Innovation. Quality. Performance. Best in Industry. Worldwide.

MultiCam® Router User Manual

Introduction	IN 1 – 9
Fast Starts	FS 1 – 50
Keypad and Functions	KF 1 – 40
Menu Items	MN 1 – 114
Options	OP 1 – 19
Connections	CN 1 – 19
Software	SW 1 – 41

Read this manual before using this product.

Failure to follow the instructions and safety precautions in this manual can result in serious injury or death.

Keep this manual in a safe location.

Introduction

top

Customer Responsibilities for Safety 2
Power Lockout Instructions 4
Labels 5

Customer Responsibilities for Safety

As the user of the ***MultiCam*** routing system, each customer **must** follow all safety procedures that are clearly outlined when operating each machine and its parts. Failure to do so may result in damage to property or personnel and could even result in death. These machines are powerful and should always be treated with care.

Customers should make plans for safeguarding themselves and their work areas at the point of operation as all ***MultiCam*** machines have been designed and constructed to operate under specific parameters relating to the particular application. As a result, ANSI B-11 *Safety Standards* states that "It shall be the responsibility of the end-user (buyer) to provide, and ensure the use of a guard, guarding device, awareness barrier, awareness device or shield..." in order to maintain the required level of protection.

MultiCam has available certain safety shields and guards applicable to our machines. Please contact our office for styles, types, and prices.

Personal Safety Precautions

All customers should become familiar with all aspects of their machines.

Any person who operates or does any maintenance on this equipment should be aware that safety procedures are an important part of the daily job. Customers and related personnel should learn how the equipment functions and be able to respect the capabilities of the machinery. Anyone working on or around the equipment should understand the potential losses associated with mishandling the machinery and its parts and be able and willing to follow all safety precautions. Sudden movement, loud noises, horseplay, etc. must be avoided, as such distractions may result in unsafe conditions.

Accidents can occur if clothing or other articles become entangled in the cutter or other moving parts of the machines. The following suggestions, if followed, will reduce the chance of having these types of accidents:

1. Wear **approved eye and hearing protection** at all times when operating the routing system.
2. Restrain long hair with a **cap or net** when near the machines.
3. Avoid wearing neckties and scarves during machine operation.
4. Avoid wearing loose fitting clothing during machine operation.
5. Avoid wearing hanging jewelry during machine operation.
6. Wear gloves only when handling sharp or hot parts.
7. Avoid operating this and any other equipment if affected by alcohol, drugs, or any other substance or condition that may decrease judgment or alertness.
8. **Observe and follow all safety signs** on the machine and in the surrounding areas.
9. Avoid placing hands on the tabletop when the spindle is turned on.
10. Lock out the incoming power supply when any type of maintenance or other work is being performed on the machine.

Work Area Safety

Fire extinguishers should always be readily accessible, and operators should always familiarize themselves with the fire prevention recommendations for each component of the CNC system. It is important to always keep the work area **clean and uncluttered**. Oil, debris, or water on the floor can cause unsafe conditions. Customers should be sure that all work areas are free of hazardous obstructions and that all tools and other equipment are returned to their proper storage place when not in use. Operators should never leave the machine unattended during the cutting sequence.

Cutter Safety

Cutting tools are very sharp and should be handled accordingly. Machine operators should inspect tools before each use and discard any defective tools. Failure to properly handle tooling can result in serious injury.

Installation Safety

An electrician must read and understand the electrical schematics prior to connecting the machine to the local power system. Connecting the wrong voltage power will void the warranty. All switches should be turned to the OFF position before power is connected. The main disconnect switch should always be locked in the OFF position if the machine is left unattended. When the machine is installed, the electrician or customer should be sure that all motors rotate in the correct direction. Introduction

Innovation. Quality. Performance. Best in Industry. Worldwide.

Power Lockout Instructions

MultiCam machines are equipped with a built-in main power lockout device. If any kind of repair work or maintenance is being performed to the machine or control cabinet, the operator should **disconnect power** from the machine **before starting work**.

1. Turn the main disconnect switch to the OFF position.

2. Flip out the plastic lockout latch.

3. Insert any padlock into one of the holes of the lockout latch. With the plastic lockout latch in this position, the switch cannot be moved to the ON position.

1

2

3

4. Throw the bulkhead switch to the OFF position.

5. Open the control cabinet.

6. Make sure the LEDs on the drives and inverters are not lit. It may take a few minutes for the light from the LEDs to fade away.

7. Make sure the incoming voltage for L1, L2, and L3 is 0v.

8. Follow the operator's standard Tag Out procedure in tagging the disconnect switch.

Introduction

Labels

Each machine produced by ***MultiCam*** includes certain cautions, warnings, dangers, and notices placed on the moving parts of the machine (e.g., gantry, carriage) as well as the stationary parts (e.g., electrical enclosure). These labels are in place to encourage employees to observe safe operating practices at all times. The text on these labels is almost always capitalized and presented in bold format. Any auxiliary equipment (e.g., vacuum pump, chiller) will include vendor-specific labels advising the customer of proper safety practices regarding those units as well. The most common labels are identified below.

Cautions

Caution labels indicate the potential for minor or moderate injury and are usually yellow with black writing. These labels are normally placed on the gantry, axis covers, and any other covers that are located at the front of the machine.

Warnings

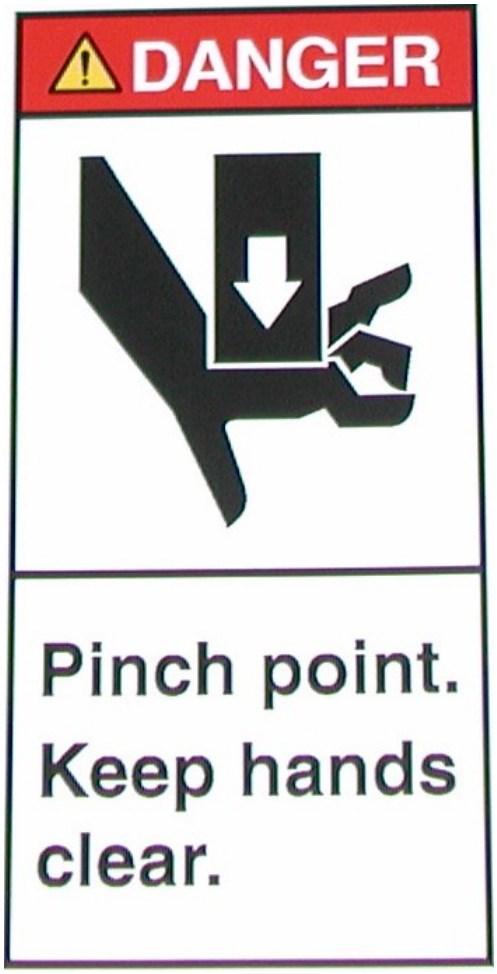
Warning labels indicate the potential for death or serious injury and usually show the word WARNING in black writing against an orange background. These labels are normally located in areas where serious injury could be sustained, such as the carriage.

WARNING
Crush hazard.
To avoid tip over, this equipment must be securely fastened to the floor.

WARNING
Enclosure is TOP–HEAVY
• Stabilize enclosure before opening doors.
• Follow guidelines in "Large Enclosure Handling Manual".
• DO NOT remove this label until enclosure is permanently installed.
• Label must be reattached if removed for painting.
• To get additional copies of the "Large Enclosure Handling Manual" or labels, call 1–800–355–3560.
93268001, Dwg. Rev. G

Dangers

Danger labels indicate the probability of death or serious injury and usually show the word DANGER in white writing against a red background. These labels are normally located in areas where serious injury could occur, such as the knife cartridge.



Notices

Notice labels indicate the potential for property damage and usually show the word NOTICE in white writing against a blue backdrop. This label is normally located at the front of the machine.

Introduction

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Starts

top

Fast Start ATC Spindle	2
Fast Start MultiVision – ATC Spindle	5
Fast Start Manual Spindle	8
Fast Start MultiVision – Manual Spindle	11
Fast Start Cartridge Knife – ATC Spindle	14
Fast Start Cartridge Knife – Manual Spindle	18
Fast Start Cartridge Knife – MultiVision	22
Fast Start Cartridge Knife System	25
Fast Start Oscillating Knife – ATC Spindle	28
Fast Start Oscillating Knife – Manual Spindle	32
Fast Start Tangential Knife – ATC Spindle	36
Fast Start Tangential Knife – Manual Spindle	40
Hints and Tips	43
Contacting MultiCam	45
Maintenance	46
Maintenance Log	48
Computerized Routing System Warranty	49
Warranty Registration List	50

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Start – ATC Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.

2. **Find Machine Hard Home**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly.
   Press Shift and Five to open the chuck and remove a tool from the Tool Changer.

3. **Calibrate New Tools**
   Follow the Tool Calibration Procedure when calibrating new tools.
   MultiCam requires that tools be calibrated when a new tool is inserted into the tool holder. The machine will store the Z-axis tool length of the new tool in memory even after the machine is turned off. Only new tools need to be calibrated.

4. **Set Home**
   Press Jog Arrows to place the **center of the bit** over the front right corner of the material.
   Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.
   Press Set Home to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.
   Press Enter to confirm the position.

5. **Turn on the Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these ***MultiCam*** features.

6. **Set Surface**

Press Jog Arrows to position the tool bit anywhere over the material to be cut.
Place the grounding collar on the ATC mounting groove in the spindle tool holder.

Press Set Surface and enter the tool number.
Place the surface block on top of the material underneath the tool.

Press Z-axis Jog Arrows to lower the Z-axis until the end of the tool is just above the surface block.

Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
Remove the grounding collar and surface block before beginning work.

7. **Set Maximum Depth** (as needed)

Press Jog Arrows to position the tool over the spoilboard or sacrifice material.
Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press Set Max Depth to set the maximum depth for the bit.

Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press Z-axis Jog Arrows to lower the spindle until the end of the tool is just above the surface block.

Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
Remove the grounding collar and surface block before beginning work.

8. **Adjust Maximum Depth** (as needed)

Press Shift and Set Max Depth .

Press Cancel to clear the current maximum depth without having to reboot the machine or press

Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

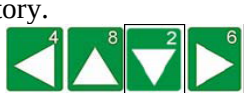
Press Enter to accept or Cancel to abort.

9. **Turn on Mister or Dust Collection Vacuum**
Make sure the manual switch on the mister is switched to the ON position if applicable.

10. **DNC**

Make sure that JobServer or JobNameServer has been activated on the computer.

Press DNC . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job folder.

Press Enter to display the DNC files that are available in that particular folder.

Press Jog Arrows until the blinking cursor is over the first letter of the appropriate job file.

Press Start to begin the file.

11. **Start / Cancel / Pause**

Press Start to begin the cutting sequence.

Press Cancel to abort the cutting sequence.

Press Pause to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Start MultiVision – ATC Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.
   Make sure that Suite 4 and VisionTool4 or MultiVision are installed on the computer
   Make sure the dongle is in place on the computer as the software will not run without the dongle.

2. **Find Machine Hard Home**
   Press Shift and Go Home to index the gantry, carriage, and Z assembly.
   Press Shift and Five to open the chuck and remove a tool from the Tool Changer.

3. **Calibrate New Tools**
   Follow the Tool Calibration Procedure when calibrating new tools.
   MultiCam requires that tools be calibrated when a new tool is inserted into the tool holder. The machine will store the Z-axis tool length of the new tool in memory even after the machine is turned off.
   Only new tools need to be calibrated.

4. **Turn on Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the surface can be set properly. Failure to turn on the vacuum will skew the configuration, which could affect the quality of the cutting sequence.

5. **Set the Surface to the Top of the Material**
   Use any calibrated tool.
   Place the grounding collar on the ATC mounting groove in the spindle tool holder.
   Press Set Surface and enter the tool number.
   Place the surface block on top of the material underneath the tool.
   Press Z-axis Jog Arrows to lower the spindle until the bit is just above the surface block. The operator should monitor the Jog speed when lowering the Z-axis as the machine will only recognize the surface block when 0 is pressed. The spindle could crash into the material or the surface block and break the bit.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the grounding collar and surface block before beginning work.

6. **Set Maximum Depth** (as needed)

Press Jog Arrows to position the tool over the spoilboard or sacrifice material.
Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press Set Max Depth to set the maximum depth for the bit.
Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press Z-axis Jog Arrows to lower the spindle until the end of the tool is just above the surface block.

Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
Remove the grounding collar and surface block before beginning work.

7. **Adjust Maximum Depth** (as needed)

Press Shift and Set Max Depth .

Press Cancel to clear the current maximum depth without having to reboot the machine or press

Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

Press Enter to accept or Cancel to abort.

8. **Set Home**

Press Jog Arrows to place the center of the bit over the first fiducial.

Press Set Home .

Press Enter to accept the location.

9. **Capture the Fiducial**

Click on Live Feed in the toolbar and Jog the camera over a fiducial. Operators may need to move the camera along the Z-axis to bring the fiducial into focus, and the spotlight will flash while the camera is in Live Feed mode.

Click on Capture Fiducial in the toolbar. A box should appear around the fiducial .

10. **Focus the Camera after Powering Up the Machine**

Press Menu

to access the menu system.

Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

Press Enter
to access the Vision sub-menu.

Press Jog Arrows

until the blinking cursor is over the “F” in Focus Camera.

Press Enter
to access the Focus Camera function. The system will prompt the operator to unload the tool.

Press Enter
to unload the tool or Cancel
to leave the tool in the spindle.

Press Jog Arrows

to reach the fiducial on the material to be cut.

Press Z-axis Jog Arrows

to focus the camera on the fiducial.

Press Enter
when the fiducial is in focus.

11. **Access a MultiVision Job File**

Press DNC
and select the file. The VisionTool application will become active and the camera will Jog to the estimated location for the first fiducial.

Watch the system Jog to the second fiducial once the camera locates the first fiducial. The value entered in Auto Find Setting will determine how far the camera will search for the second fiducial.

Watch the system interpolate the exact location of the third fiducial and scan it.

12. **Start / Cancel / Pause**

Press Start
to begin the cutting sequence.

Press Cancel
to abort the cutting sequence.

Press Pause
to yield the cutting sequence.
Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Start – Manual Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.

2. **Find Machine Hard Home**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly. The activation of this function enables the operator to access the spindle inverter.

3. **Insert Cutter**
   Insert the tool into the collet.
   Twist the tool to lock it into place.

4. **Set Home**
   Press Jog Arrows to place the **center of the bit** over the front right corner of the material.
   Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.
   Press Set Home to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.
   Press Enter to confirm the position.

5. **Turn on Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these ***MultiCam*** features.

6. **Set Surface**
   Press Set Surface .
   Place the surface block on top of the material underneath the tool.
   Press Z-axis Jog Arrows to lower the Z-axis until the end of the tool is just above the surface block.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the surface block before beginning work.

7. **Set Maximum Depth** (as needed)

   Press Jog Arrows to position the tool over the spoilboard or sacrifice material.

   Press Set Max Depth to set the maximum depth for the bit.
   Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

   Press Z-axis Jog Arrows to lower the spindle until the end of the tool is just above the surface block.

   Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the surface block before beginning work.

8. **Adjust Maximum Depth** (as needed)

   Press Shift and Set Max Depth .

   Press Cancel to clear the current maximum depth without having to reboot the machine or press

   Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

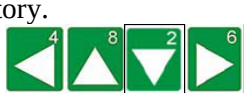
   Press Enter to accept or Cancel to abort.

9. **Turn on Mister or Dust Collection Vacuum**
   Verify the manual switch on the mister is switched to the ON position if applicable.

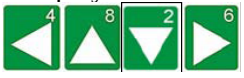
10. **DNC**

    Verify that JobServer or JobNameServer has been activated on the computer.

    Press DNC . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job folder.

    Press Enter to display the DNC files that are available in that particular folder.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job file.

    Press Start to begin the file.

11. **Start / Cancel / Pause**

Press Start to begin the cutting sequence.

Press Cancel to abort the cutting sequence.

Press Pause to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Start MultiVision – Manual Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.
   Make sure that Suite 4 and VisionTool4 or MultiVision are installed on the computer
   Make sure the dongle is in place on the computer as the software will not run without the dongle.

2. **Find Machine Hard Home**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly. The activation of this function enables the operator to access the spindle inverter.

3. **Insert Cutter**
   Insert the tool into the collet.
   Twist the tool to lock it into place.

4. **Turn on Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the surface and maximum depth can be set properly. Failure to turn on the vacuum will skew the configuration, which could affect the quality of the cutting sequence.

5. **Set the Surface to the Top of the Material**
   Press Set Surface .
   Place the surface block on top of the material underneath the tool.
   Press Z-axis Jog Arrows to lower the spindle until the bit is just above the surface block. The operator should monitor the Jog speed when lowering the Z-axis as the machine will only recognize the surface block when 0 is pressed. The spindle could crash into the material or the surface block and break the bit.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the surface block before beginning work.

6. **Set Maximum Depth** (as needed)

   Press Jog Arrows to position the tool over the spoilboard or sacrifice material.

   Press Set Max Depth to set the maximum depth for the bit.
   Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

   Press Z-axis Jog Arrows to lower the spindle until the end of the tool is just above the surface block.

   Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the surface block before beginning work.

7. **Adjust Maximum Depth** (as needed)

   Press Shift and Set Max Depth .

   Press Cancel to clear the current maximum depth without having to reboot the machine or press

   Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

   Press Enter to accept or Cancel to abort.

8. **Set Home**

   Press Jog Arrows to place the center of the bit over the first fiducial.

   Press Set Home .

   Press Enter to accept the location.

9. **Capture the Fiducial**

   Click on Live Feed in the toolbar and Jog the camera over a fiducial. Operators may need to move the camera along the Z-axis to bring the fiducial into focus, and the spotlight will flash while the camera is in Live Feed mode.

   Click on Capture Fiducial in the toolbar. A box should appear around the fiducial .

10. **Focus the Camera Each Time the Material Thickness Changes**

Press Menu

to access the menu system.

Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

Press Enter
to access the Vision sub-menu.

Press Jog Arrows

until the blinking cursor is over the “F” in Focus Camera.

Press Enter
to access the Focus Camera function. The system will prompt the operator to unload the tool.

Press Enter
to unload the tool or Cancel
to leave the tool in the spindle.

Press Jog Arrows

to reach the fiducial on the material to be cut.

Press Z-axis Jog Arrows

to focus the camera on the fiducial.

Press Enter
when the fiducial is in focus.

11. **Access a MultiVision Job File**

Press DNC
and select the file. The VisionTool application will become active and the camera will Jog to the estimated location for the first fiducial.
Watch the system Jog to the second fiducial once the camera locates the first fiducial. The value entered in Auto Find Setting will determine how far the camera will search for the second fiducial.
Watch the system interpolate the exact location of the third fiducial and scan it.

12. **Start / Cancel / Pause**

Press Start
to begin the cutting sequence.

Press Cancel
to abort the cutting sequence.

Press Pause
to yield the cutting sequence.
Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Start Cartridge Knife – ATC Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.

2. **Find Machine Hard Home with the Router**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly.
   Press Shift and Five to open the chuck and remove a tool from the Tool Changer.

3. **Calibrate New Tools** (including Cartridges 31 – 40 and 41 – 50 if applicable)
   Follow the Tool Calibration Procedure when calibrating new tools.
   MultiCam requires that tools be calibrated when a new tool is inserted into the tool holder. The machine will store the Z-axis tool length of the new tool in memory even after the machine is turned off.
   Only new tools need to be calibrated.
   Remove the grounding collar after each tool is calibrated. No grounding collar is needed when calibrating the knife.

4. **Turn on the Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these ***MultiCam*** features.

5. **Set Surface with the Router** (must be reset each time the machine is turned on, the bit changes, or the material thickness changes)
   Press Jog Arrows to position any calibrated tool anywhere over the surface block. If operators are calibrating the knife, then they should set surface with Tool 1.
   Place the grounding collar on the ATC mounting groove in the spindle tool holder.
   Press Set Surface and enter the tool number.
   Place the surface block on top of the material underneath the tool.
   Press Z-axis Jog Arrows to lower the Z-axis until the end of the tool is just above the surface block.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

6. **Set Maximum Depth for the Router** (as needed and then after each reboot)

   Press Jog Arrows to position the tool over the spoilboard or sacrifice material.
   Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

   Press Set Max Depth to set the maximum depth for the bit.

   Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

   Press Z-axis Jog Arrows to lower the spindle until the end of the tool is just above the surface block.

   Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height. **Maximum Depth** has no effect on the knife.
   Remove the grounding collar and surface block before beginning work.

7. **Adjust Maximum Depth for the Router** (as needed)

   Press Shift and Set Max Depth .

   Press Cancel to clear the current maximum depth without having to reboot the machine or press

   Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

   Press Enter to accept or Cancel to abort. **Adjust Maximum Depth** has no effect on the knife.

8. **Set Home with the Router**

   Press Jog Arrows to place the **center of the bit** over the front right corner of the material.

   Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

   Press Set Home to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

   Press Enter to confirm the position.

9. **Set Knife Params**

Job File

Press Menu .

Press Jog Arrows until the blinking cursor is over the “K” in KnifeUtil.

Press Enter to access the Knife Utility menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife Z Control.

Press Enter to toggle to Job.

Skip to 10. **Turn on Mister or Dust Collection Vacuum** below.

Keypad

Press Menu .

Press Jog Arrows until the blinking cursor is over the “K” in KnifeUtil.

Press Enter to access the Knife Utility menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife Z Control.

Press Enter to toggle to Keypad.

Press Cancel to exit the KnifeUtil sub-menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife.

Press Enter to access the Knife menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife Depth.

Press Enter to set the knife depth.

Enter a positive or negative value for the knife depth and press Enter to return to the Knife menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife Lift.

Press Enter to set the knife lift.

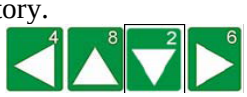
Enter the value for the knife lift and press Enter to return to the Knife menu.

10. **Turn on Mister or Dust Collection Vacuum**
    Make sure the manual switch on the mister is switched to the ON position if applicable.

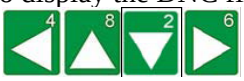
11. **DNC**

    Make sure that JobNameServer has been activated on the computer.

    Press DNC . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job folder.

    Press Enter to display the DNC files that are available in that particular folder.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job file.

    Press Start to begin the file.

12. **Start / Cancel / Pause**

    Press Start to begin the cutting sequence.

    Press Cancel to abort the cutting sequence.

    Press Pause to yield the cutting sequence.
    Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Start Cartridge Knife – Manual Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.

2. **Find Machine Hard Home with the Router**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly.
   Press Shift and Five to open the chuck and remove a tool from the Tool Changer.

3. **Insert Cutter**
   Insert the tool into the collet.
   Twist the tool to lock it into place.

4. **Turn on the Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these ***MultiCam*** features.

5. **Set Home with the Router**
   Press Jog Arrows to place the **center of the bit** over the front right corner of the material.
   Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.
   Press Set Home to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.
   Press Enter to confirm the position.

6. **Set Surface with the Router**
   Press Set Surface and enter any regular tool number (e.g., 1).
   Place the surface block on top of the material underneath the tool.
   Press Z-axis Jog Arrows to lower the Z-axis until the end of the tool is just above the surface block.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the surface block before beginning work.

7. **Set Surface with the Knife**

Press Set Surface and enter any knife tool number (e.g., 31).

Press Cancel to leave oscillations inactive. Oscillation stroke can be added in later.
Place the surface block on top of the material underneath the tool.

Press Z-axis Jog Arrows to lower the Z-axis until the end of the knife is just above the surface block.

Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Press Enter to add in oscillating stroke or Cancel to exclude oscillating stroke.
Remove the surface block before beginning work.

8. **Set Maximum Depth for the Router** (as needed and then after each reboot)

Press Jog Arrows to position the tool over the spoilboard or sacrifice material.
Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press Set Max Depth to set the maximum depth for the bit.

Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press Z-axis Jog Arrows to lower the spindle until the end of the tool is just above the surface block.

Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height. **Maximum Depth** has no effect on the knife.
Remove the grounding collar and surface block before beginning work.

9. **Adjust Maximum Depth for the Router** (as needed)

Press Shift and Set Max Depth .

Press Cancel to clear the current maximum depth without having to reboot the machine or press

Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

Press Enter to accept or Cancel to abort. **Adjust Maximum Depth** has no effect on the knife.

10. **Set Knife Params**

Job File

Press Menu .

Press Jog Arrows until the blinking cursor is over the “K” in KnifeUtil.

Press Enter to access the Knife Utility menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife Z Control.

Press Enter to toggle to Job.

Skip to 10. **Turn on Mister or Dust Collection Vacuum** below.

Keypad

Press Menu .

Press Jog Arrows until the blinking cursor is over the “K” in KnifeUtil.

Press Enter to access the Knife Utility menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife Z Control.

Press Enter to toggle to Keypad.

Press Cancel to exit the KnifeUtil sub-menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife.

Press Enter to access the Knife menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife Depth.

Press Enter to set the knife depth.

Enter a positive or negative value for the knife depth and press Enter to return to the Knife menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife Lift.

Press Enter to set the knife lift.

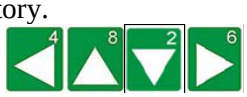
Enter the value for the knife lift and press Enter to return to the Knife menu.

11. **Turn on Mister or Dust Collection Vacuum**
    Make sure the manual switch on the mister is switched to the ON position if applicable.

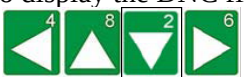
12. **DNC**

    Make sure that JobNameServer has been activated on the computer.

    Press DNC . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job folder.

    Press Enter to display the DNC files that are available in that particular folder.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job file.

    Press Start to begin the file.

13. **Start / Cancel / Pause**

    Press Start to begin the cutting sequence.

    Press Cancel to abort the cutting sequence.

    Press Pause to yield the cutting sequence.
    Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Fast Start Cartridge Knife– MultiVision

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Make sure that Suite 4 and VisionTool4 or MultiVision are installed on the computer
   Make sure the dongle is in place on the computer as the software will not run without the dongle.

2. **Find Machine Hard Home**
   Press Shift and Go Home to index the gantry, carriage, and Z assembly.

3. **Insert Cartridge**
   Insert the cartridge into the cartridge receiver.
   Tighten the 3 bolts. The cartridge will only fit inside the cartridge receiver one way.

4. **Turn on Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the surface and depth can be set properly. Failure to turn on the vacuum will skew the configuration, which could affect the quality of the cutting sequence.

5. **Set Home**
   Press Jog Arrows to place the **center of the blade** over the front right corner of the material.

   Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

   Press Set Home to set the home location. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

   Press Enter to confirm the position.

6. **Set Knife Params**

Press Menu

.

Press Jog Arrows

until the blinking cursor is over the “K” in KnifeUtil.

Press Enter to access the Knife Utility menu.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife Z Control.

Press Enter to toggle to Keypad.

Press Cancel to exit the KnifeUtil sub-menu.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife.

Press Enter to access the Knife menu.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife Depth.

Press Enter to set the knife depth.

Enter a positive or negative value for the knife depth and press Enter to return to the Knife menu.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife Lift.

Press Enter to set the knife lift.

Enter the value for the knife lift and press Enter to return to the Knife menu.

7. **Set Surface with the Knife** (must be reset each time the machine is turned on, the blade changes, or the material thickness changes)

Press Set Surface and enter any knife tool number (e.g., 31).

Press Cancel to leave oscillations inactive. Oscillation stroke can be added in later.
Place the surface block on top of the material underneath the tool.

Press Z-axis Jog Arrows

to lower the Z-axis until the end of the knife is just above the surface block.
On Solenoid-controlled systems, the surface must be set to the bottom of the material.

Press and hold Zero

to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Knife Lift Height.

Press Enter to add in oscillating stroke or Cancel to exclude oscillating stroke.
Remove the surface block before beginning work.

8. **Capture the Fiducial**

   Click on Live Feed in the toolbar and Jog the camera over a fiducial. Operators may need to move the camera along the Z-axis to bring the fiducial into focus, and the spotlight will flash while the camera is in Live Feed mode.

   Click on Capture Fiducial in the toolbar. A box should appear around the fiducial .

9. **Focus the Camera Each Time the Material Thickness Changes**

   Press Menu to access the menu system.

   Press Jog Arrows until the blinking cursor is over the “V” in Vision.

   Press Enter to access the Vision sub-menu.

   Press Jog Arrows until the blinking cursor is over the “F” in Focus Camera.

   Press Enter to access the Focus Camera function.

   Press Jog Arrows to reach the fiducial on the material to be cut.

   Press Z-axis Jog Arrows to focus the camera on the fiducial.

   Press Enter when the fiducial is in focus.

10. **Access a MultiVision Job File**

    Press DNC and select the file. The VisionTool application will become active and the camera will Jog to the estimated location for the first fiducial.

    Watch the system Jog to the second fiducial once the camera locates the first fiducial. The value entered in Auto Find Setting will determine how far the camera will search for the second fiducial.

    Watch the system interpolate the exact location of the third fiducial and scan it.

11. **Start / Cancel / Pause**

    Press Start to begin the cutting sequence.

    Press Cancel to abort the cutting sequence.

    Press Pause to yield the cutting sequence.
    Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Start Cartridge Knife System

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.

2. **Find Machine Hard Home**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly.
   Press Shift and Five to open the chuck and remove a tool from the Tool Changer.

3. **Insert Cartridge**
   Insert the cartridge into the cartridge receiver.
   Tighten the 3 bolts. The cartridge will only fit inside the cartridge receiver one way.

4. **Turn on the Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the **surface and depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these ***MultiCam*** features.

5. **Set Home**
   Press Jog Arrows to place the **center of the blade** over the front right corner of the material.
   Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.
   Press Set Home to set the home location. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.
   Press Enter to confirm the position.

6. **Set Knife Params**

Press Menu

.

Press Jog Arrows

until the blinking cursor is over the “K” in KnifeUtil.

Press Enter to access the Knife Utility menu.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife Z Control.

Press Enter to toggle to Keypad.

Press Cancel to exit the KnifeUtil sub-menu.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife.

Press Enter to access the Knife menu.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife Depth.

Press Enter to set the knife depth.

Enter a positive or negative value for the knife depth and press Enter to return to the Knife menu.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife Lift.

Press Enter to set the knife lift.

Enter the value for the knife lift and press Enter to return to the Knife menu.

7. **Set Surface with the Knife** (must be reset each time the machine is turned on, the blade changes, or the material thickness changes)

Press Set Surface and enter any knife tool number (e.g., 31).

Press Cancel to leave oscillations inactive. Oscillation stroke can be added in later.
Place the surface block on top of the material underneath the tool.

Press Z-axis Jog Arrows to lower the Z-axis until the end of the knife is just above the surface block.

Press and hold Zero

to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Knife Lift Height.

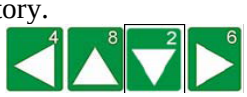
Press Enter to add in oscillating stroke or Cancel to exclude oscillating stroke.
Remove the surface block before beginning work.

8. **Turn on Mister or Dust Collection Vacuum**
   Make sure the manual switch on the mister is switched to the ON position if applicable.

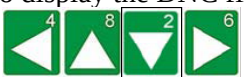
9. **DNC**

   Make sure that JobNameServer has been activated on the computer.

   Press DNC . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

   Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job folder.

   Press Enter to display the DNC files that are available in that particular folder.

   Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job file.

   Press Start to begin the file.

10. **Start / Cancel / Pause**

    Press Start to begin the cutting sequence.

    Press Cancel to abort the cutting sequence.

    Press Pause to yield the cutting sequence.
    Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Start Oscillating Knife – ATC Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.

2. **Find Machine Hard Home with the Router**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly.
   Press Shift and Five to open the chuck and remove a tool from the Tool Changer.

3. **Calibrate New Tools** (must calibrate and surface tool 1 before calibrating knife)
   Follow the Tool Calibration Procedure when calibrating new tools.
   MultiCam requires that tools be calibrated when a new tool is inserted into the tool holder. The machine will store the Z-axis tool length of the new tool in memory even after the machine is turned off. Only new tools need to be calibrated.

4. **Turn on the Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these ***MultiCam*** features.

5. **Set Surface with the Router** (must be reset each time the machine is turned on, the bit changes, or the material thickness changes)
   Press Jog Arrows to position any calibrated tool anywhere over the surface block. If operators are calibrating the knife, then they should set surface with Tool 1.
   Place the grounding collar on the ATC mounting groove in the spindle tool holder.
   Press Set Surface and enter the tool number.
   Place the surface block on top of the material underneath the tool.
   Press Z-axis Jog Arrows to lower the Z-axis until the end of the tool is just above the surface block.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the grounding collar and surface block before beginning work.

6. **Calibrate the Knife** (only if the blade has been replaced)

Press Menu

.

Press Jog Arrows

until the blinking cursor is over the “A” in ATC.

Press Enter to access the ATC sub-menu.

Press Jog Arrows

until the blinking cursor is over the “C” in Cal_Tool.

Press Enter to begin calibration.

Enter the Tool Number of the appropriate knife. Knives are arranged as tools 61, 62, or 63 depending on the number of knives on the machine and their location from the spindle.

Press Enter . The display will prompt the operator to identify the knife as oscillating or not.

Press Enter to indicate the knife is oscillating or Cancel to indicate the knife is not oscillating.

7. **Set Surface with the Oscillating or Non-oscillating Knife as controlled in Parameter 372** (set knife depth to a negative value to prevent the oscillation from going too deep in the material)

Set Surface with the oscillating knife.

--Press Set Surface and enter the tool number (e.g., 61 for Knife 1).

--Press Enter to select Oscillating.

--Press Z-axis Jog Arrows

until the blade just penetrates the top of the material.

--Press Enter to set surface.

Set Surface with the non-oscillating knife. The stroke of the knife has not been configured in the surface setting procedure for a non-oscillating knife, which means that the knife will cut into the material by the stroke of the oscillation if the knife depth is 0.0.

--Press Set Surface and enter the tool number (e.g., 61 for Knife 1).

--Press Cancel to select Non-oscillating

--Place the surface block on the spoilboard underneath the knife.

--Press Z-axis Jog Arrows

to bring the blade closer to the surface block.

--Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the knife comes in contact with the surface block and then move up above the surface block to the Knife Lift Height.

--Remove the surface block before beginning work.

8. **Set Knife Depth**

Press Menu

.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife.

Press Enter to access the Knife menu.

Press Jog Arrows

until the blinking cursor is over the “K” in Knife Depth.

Press Enter to set the knife depth.
Enter a positive or negative value for the knife depth. Operators of non-oscillating knives may prefer a negative knife depth.

9. **Set Home with the Router**

Press Jog Arrows

to place the **center of the bit** over the front right corner of the material.

Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

Press Set Home to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

Press Enter to confirm the position.

10. **Set Maximum Depth for the Router** (as needed and then after each reboot)

Press Jog Arrows

to position the tool over the spoilboard or sacrifice material.
Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press Set Max Depth to set the maximum depth for the bit.

Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press Z-axis Jog Arrows

to lower the spindle until the end of the tool is just above the surface block.

Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height. **Maximum Depth has no effect on the knife**.
Remove the grounding collar and surface block before beginning work.

11. **Adjust Maximum Depth for the Router** (as needed)

    Press Shift and Set Max Depth .

    Press Cancel to clear the current maximum depth without having to reboot the machine or press Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

    Press Enter to accept or Cancel to abort. **Adjust Maximum Depth has no effect on the knife**.

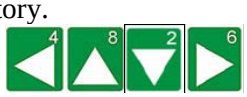
12. **Turn on Mister or Dust Collection Vacuum**

    Make sure the manual switch on the mister is switched to the ON position if applicable.

13. **DNC**

    Make sure that JobServer or JobNameServer has been activated on the computer.

    Press DNC . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job folder.

    Press Enter to display the DNC files that are available in that particular folder.

    Press Jog Arrows until the blinking cursor is over the first letter of the appropriate job file.

    Press Start to begin the file.

14. **Start / Cancel / Pause**

    Press Start to begin the cutting sequence.

    Press Cancel to abort the cutting sequence.

    Press Pause to yield the cutting sequence.

    Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Fast Start Oscillating Knife – Manual Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.

2. **Find Machine Hard Home with the Router**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly. The activation of this function enables the operator to access the spindle inverter.

3. **Turn on the Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these ***MultiCam*** features.

4. **Set Surface with the Router** (must be reset each time the machine is turned on, the bit changes, or the material thickness changes)
   Press Jog Arrows to position Tool 1 anywhere over the surface block.
   Place the grounding collar on the ATC mounting groove in the spindle tool holder.
   Press Set Surface and enter the tool number.
   Place the surface block on top of the material underneath the tool.
   Press Z-axis Jog Arrows to lower the Z-axis until the end of the tool is just above the surface block.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the grounding collar and surface block before beginning work.

5. **Set Surface with the Oscillating or Non-oscillating Knife as controlled in Parameter 372** (must be reset each time the machine is turned on or the blade is changed)

   Set Surface with the oscillating knife.

   --Press Set Surface and enter the tool number (e.g., 61 for Knife 1).

   --Press Enter to select Oscillating.

   --Press Z-axis Jog Arrows until the blade just penetrates the top of the material.

   --Press Enter to set surface.

Set Surface with the non-oscillating knife. The stroke of the knife has not been configured in the surface setting procedure for a non-oscillating knife, which means that the knife will cut into the material by the stroke of the oscillation if the knife depth is 0.0.

--Press Set Surface and enter the tool number (e.g., 61 for Knife 1).

--Press Cancel to select Non-oscillating
--Place the surface block on the spoilboard underneath the knife.

--Press Z-axis Jog Arrows to bring the blade closer to the surface block.

--Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the knife comes in contact with the surface block and then move up above the surface block to the Knife Lift Height.
--Remove the surface block before beginning work.

6. **Set Knife Depth**

Press Menu .

Press Jog Arrows until the blinking cursor is over the “K” in Knife.

Press Enter to access the Knife menu.

Press Jog Arrows until the blinking cursor is over the “K” in Knife Depth.

Press Enter to set the knife depth.
Enter a positive or negative value for the knife depth. Operators of non-oscillating knives may prefer a negative knife depth.

7. **Set Home with the Router**

Press Jog Arrows to place the **center of the bit** over the front right corner of the material.

Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

Press Set Home to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

Press Enter to confirm the position.

8. **Set Maximum Depth for the Router** (as needed and then after each reboot)

   Press Jog Arrows to position the tool over the spoilboard or sacrifice material.
   Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

   Press Set Max Depth to set the maximum depth for the bit.
   Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

   Press Z-axis Jog Arrows to lower the spindle until the end of the tool is just above the surface block.

   Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height. **Maximum Depth has no effect on the knife**.
   Remove the grounding collar and surface block before beginning work.

9. **Adjust Maximum Depth for the Router** (as needed)

   Press Shift and Set Max Depth .

   Press Cancel to clear the current maximum depth without having to reboot the machine or press

   Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

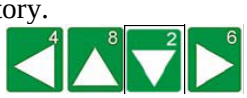
   Press Enter to accept or Cancel to abort. **Adjust Maximum Depth has no effect on the knife**.

10. **Turn on Mister or Dust Collection Vacuum**
    Make sure the manual switch on the mister is switched to the ON position if applicable.

11. **DNC**

    Make sure that JobServer or JobNameServer has been activated on the computer.

    Press DNC . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job folder.

    Press Enter to display the DNC files that are available in that particular folder.

    Press Jog Arrows until the blinking cursor is over the first letter of the appropriate job file.

    Press Start to begin the file.

12. **Start / Cancel / Pause**

Press Start to begin the cutting sequence.

Press Cancel to abort the cutting sequence.

Press Pause to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Start Tangential Knife – ATC Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.

2. **Find Machine Hard Home with the Router**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly.
   Press Shift and Five to open the chuck and remove a tool from the Tool Changer.

3. **Calibrate New Tools** (must calibrate and surface tool 1 before calibrating knife)
   Follow the Tool Calibration Procedure when calibrating new tools.
   MultiCam requires that tools be calibrated when a new tool is inserted into the tool holder. The machine will store the Z-axis tool length of the new tool in memory even after the machine is turned off. Only new tools need to be calibrated.

4. **Turn on the Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these ***MultiCam*** features.

5. **Set Surface with the Router** (must be reset each time the machine is turned on, the bit changes, or the material thickness changes)
   Press Jog Arrows to position any calibrated tool anywhere over the surface block. If operators are calibrating the knife, then they should set surface with Tool 1.
   Place the grounding collar on the ATC mounting groove in the spindle tool holder.
   Press Set Surface .
   Place the surface block on top of the material underneath the tool.
   Press Z-axis Jog Arrows to lower the Z-axis until the end of the tool is just above the surface block.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the grounding collar and surface block before beginning work.

6. **Calibrate the Knife** (only if the blade has been replaced)

   Press Menu

.

   Press Jog Arrows

until the blinking cursor is over the “A” in ATC.

   Press Enter to access the ATC sub-menu.

   Press Jog Arrows

until the blinking cursor is over the “C” in Cal_Tool.

   Press Enter to begin calibration.

   Enter the Tool Number of the appropriate knife. Knives are arranged as tools 61, 62, or 63 depending on the number of knives on the machine and their location from the spindle.

7. **Set Surface with the Knife** (must be set to a value slightly greater than the thickness of material in order to cut through)

   Press Set Surface and enter the tool number (e.g., 61 for Knife 1).

   Place the surface block on the spoilboard underneath the knife.

   Press Z-axis Jog Arrows

to bring the blade closer to the surface block.

   Press and hold Zero

to begin the surfacing routine. This will cause the Z-axis to move down slowly until the knife comes in contact with the surface block and then move up above the surface block to the Knife Lift Height.

   Remove the surface block before beginning work.

8. **Set Knife Depth**

   Press Menu

.

   Press Jog Arrows

until the blinking cursor is over the “K” in Knife.

   Press Enter to access the Knife menu.

   Press Jog Arrows

until the blinking cursor is over the “K” in Knife Depth.

   Press Enter to set the knife depth.

   Enter a positive or negative value for the knife depth. Operators of non-oscillating knives may prefer a negative knife depth.

9. **Set Home with the Router**

   Press Jog Arrows to place the **center of the bit** over the front right corner of the material.

   Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

   Press Set Home to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

   Press Enter to confirm the position.

10. **Set Maximum Depth for the Router** (as needed and then after each reboot)

    Press Jog Arrows to position the tool over the spoilboard or sacrifice material.
    Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

    Press Set Max Depth to set the maximum depth for the bit.

    Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

    Press Z-axis Jog Arrows to lower the spindle until the end of the tool is just above the surface block.

    Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height. **Maximum Depth has no effect on the knife**.
    Remove the grounding collar and surface block before beginning work.

11. **Adjust Maximum Depth for the Router** (as needed)

    Press Shift and Set Max Depth .

    Press Cancel to clear the current maximum depth without having to reboot the machine or press

    Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

    Press Enter to accept or Cancel to abort. **Adjust Maximum Depth has no effect on the knife**.

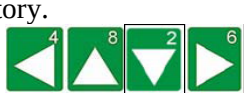
12. **Turn on Mister or Dust Collection Vacuum**

    Make sure the manual switch on the mister is switched to the ON position if applicable.

13. **DNC**

Make sure that JobServer or JobNameServer has been activated on the computer.

Press DNC . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job folder.

Press Enter to display the DNC files that are available in that particular folder.

Press Jog Arrows until the blinking cursor is over the first letter of the appropriate job file.

Press Start to begin the file.

14. **Start / Cancel / Pause**

Press Start to begin the cutting sequence.

Press Cancel to abort the cutting sequence.

Press Pause to yield the cutting sequence.
Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Fast Start Tangential Knife – Manual Spindle

1. **Turn on Power**
   Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
   Turn on the air regulator.

2. **Find Machine Hard Home with the Router**
   Press Shift and Go Home on the keypad to index the gantry, carriage, and Z assembly. The activation of this function enables the operator to access the spindle inverter.

3. **Turn on the Table Vacuum**
   Locate the power switch on the table vacuum.
   Switch the indicator to the ON position.
   The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these ***MultiCam*** features.

4. **Set Surface with the Router** (must be reset each time the machine is turned on, the bit changes, or the material thickness changes)
   Press Jog Arrows to position Tool 1 anywhere over the surface block.
   Place the grounding collar on the ATC mounting groove in the spindle tool holder.
   Press Set Surface and enter the tool number.
   Place the surface block on top of the material underneath the tool.
   Press Z-axis Jog Arrows to lower the Z-axis until the end of the tool is just above the surface block.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
   Remove the grounding collar and surface block before beginning work.

5. **Set Surface with the Knife**
   Press Set Surface and enter the tool number (e.g., 61 for Knife 1).
   Place the surface block on the spoilboard underneath the knife.
   Press Z-axis Jog Arrows to bring the blade closer to the surface block.
   Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the knife comes in contact with the surface block and then move up above the surface block to the Knife Lift Height.
   Remove the surface block before beginning work.

6. **Set Knife Depth** (must be set to a value slightly greater than the thickness of material in order to cut through)

   Press Menu

.

   Press Jog Arrows

until the blinking cursor is over the “K” in Knife.

   Press Enter to access the Knife menu.

   Press Jog Arrows

until the blinking cursor is over the “K” in Knife Depth.

   Press Enter to set the knife depth.

7. **Set Home with the Router**

   Press Jog Arrows

to place the **center of the bit** over the front right corner of the material.

   Press Jog Speed to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

   Press Set Home to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

   Press Enter to confirm the position.

8. **Set Maximum Depth for the Router** (as needed and then after each reboot)

   Press Jog Arrows

to position the tool over the spoilboard or sacrifice material.
   Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

   Press Set Max Depth to set the maximum depth for the bit.
   Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

   Press Z-axis Jog Arrows

to lower the spindle until the end of the tool is just above the surface block.

   Press and hold Zero

to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height. **Maximum Depth has no effect on the knife**.
   Remove the grounding collar and surface block before beginning work.

9. **Adjust Maximum Depth for the Router** (as needed)

   Press Shift and Set Max Depth .

   Press Cancel to clear the current maximum depth without having to reboot the machine or press Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

   Press Enter to accept or Cancel to abort. **Adjust Maximum Depth has no effect on the knife**.

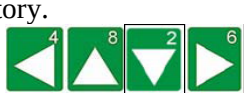
10. **Turn on Mister or Dust Collection Vacuum**

    Make sure the manual switch on the mister is switched to the ON position if applicable.

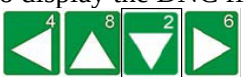
11. **DNC**

    Make sure that JobServer or JobNameServer has been activated on the computer.

    Press DNC . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job folder.

    Press Enter to display the DNC files that are available in that particular folder.

    Press Jog Arrows  until the blinking cursor is over the first letter of the appropriate job file.

    Press Start to begin the file.

12. **Start / Cancel / Pause**

    Press Start to begin the cutting sequence.

    Press Cancel to abort the cutting sequence.

    Press Pause to yield the cutting sequence.

    Operators should never leave the machine unattended during the cutting sequence.

Fast Starts

Hints and Tips

The ***MultiCam*** systems have a very powerful yet easy-to-use interface. Operators who understand their systems and learn the keypad functions can utilize their cutting systems in an efficient and productive manner. The following tips complement this knowledge and can be used to the operators' advantage. Any operator with additional helpful hints or tips should send an email to support@multicam.com so that the User Manual can be updated.

#1 Maintaining Safety

Safety glasses and hearing protection **should be worn at all times** while operating the machine, and long hair should be **restrained with a cap or net** when near the machines. Operators should never leave the machine unattended during the cutting sequence, and labels should be observed at all times.

#2 Testing

When testing new processes or when learning new functions of the machine, use the following 3 procedures:

1. Insert a pen into the end of the spindle and run the file on a piece of paper. Disable the spindle or the pen will spin.

2. Set an imaginary surface above the material high enough that the spindle will still move up and down but never come in contact with the material.

3. Use the Dry Run function.

Do **not** let the spindle run.

#3 Recovering after Loss of Power

When Set Home (0,0) is pressed, the X,Y locations are stored in the controller's memory. If the system loses power, the job can be restored by returning to the last home and running Proximity Restart.

Power can be applied back to the machine by pressing Find Home to index the machine. When the machine has completed the find home function, the operator should press Return to Home 0 to move the machine back to the last home position and set home there before re-establishing the Set Surface (Z=0), Cut Depth, Tool Lift, Cut Speed, and Z Plunge Feed. The Proximity Restart function can then be used to resume the file.

#4 Cutting Metal

Single flute spiral up-cutters should be used when cutting aluminum or brass since they allow for faster feedrates and longer cutter life. These cutters will cut out bigger chips, a process which removes heat from the cut and reduces welding. Two (2) flute cutters generally produce a slightly better edge finish than single flute cutters.

Soft metals, such as 30-31 aluminum, should be cut at a high feedrate with single flute straight or spiral cutters.

Three (3) flute Titanium Nitrate (TIN) coated bits should be used for cutting stainless steel.

The Micro Drop Misting System or Coolubricator System should always be used when cutting metals.

#5 Cutting Plastics

The Pressure Foot and the Micro Drop Misting System or Coolubricator System can be very useful tools when cutting plastics. The pressure foot can be used to remove chips and keep the chips from welding back into the cut, and the mister can keep chips from welding back into the cut and help polish the cut.

#6 Maintaining the Machine

The ***MultiCam*** Router will produce consistent, accurate work as long as it is well-maintained. Maintenance for this system is easy, yet it should not be overlooked. Operators may refer to the maintenance chart for more detail.

All machines should be kept clean when not in use. The build-up of chips and dust on the system should be cleared daily with special attention focused on the gear rack and bearing rails.

All of the bearings should be greased at least twice a month based on an 8-hour work day. The bearings should be greased more often if the machine is in use more than 8 hours a day.

The Lead Screw Z-axis should be lubricated with a non-aerosol, silicon-based lubricant. Operators are cautioned **against** using the lithium grease provided by ***MultiCam*** when lubricating Lead Screws as this may lead to premature wear of the Lead Screw. Ball Screws, however, can be lubricated with lithium grease.

The filters on the electronics enclosure and vacuum pump should also be cleaned.

#7 Cutting to Paper Mask

When cutting plastics or paper-masked materials, operators should set the maximum depth to allow for cutting through the material without cutting the paper on the back.

When setting up the system for this application, operators should place a piece of paper under the calibration block. This will set the max depth on top of the paper. Fast Starts

Contacting *MultiCam*

For Sales questions or concerns, please direct all necessary email correspondence to the following address:

sales@multicam.com

Customers with technical questions about their ***MultiCam*** equipment should refer to their User Manual first. Any caller with questions regarding an issue that is clearly defined in the manual will be directed back to the manual by the Service Technician.

If the customer is unable to find the answers in the User Manual, he or she may contact the local ***MultiCam*** distributor. If the customer still has questions, he or she may contact ***MultiCam*** by phone, fax, or email.

When placing a service call, customers should include the model number of the router, serial number of the machine, and full name of the company.

MultiCam Headquarters
1025 West Royal Lane
DFW Airport, TX 75261
USA

Mailing Address:
PO Box 612048
Dallas, TX 75261
USA

Phone: (972) 929-4070
Fax: (972) 929-4071
www.MultiCam.com

Fast Starts

Maintenance

The ***MultiCam*** Router will provide years of productive service if it is maintained properly. There are daily, weekly, monthly, quarterly, and yearly maintenance steps required for each machine based on a 40-hour work week. ***MultiCam*** recommends that any routing system used in excess of 40 hours a week should have the maintenance schedule adjusted accordingly.

Daily

The ***MultiCam*** Router should be cleaned off each day, and the water lines should be inspected. All chips should be blown or vacuumed off the machine. Close attention should be paid to the rack area since chip build-up in the rack can cause stalling problems. Customers who operate systems with a Lead Screw Z-axis should lubricate the screw with a non-aerosol, silicon-based lubricant. Operators should **not** use WD-40 on the Z-axis screw for lubrication. Using anything other than what is recommended will result in the premature wear of the Z-axis screw. With Ball Screw systems, oil or lithium grease is an acceptable lubricant.

Weekly

The ***MultiCam*** Router should be cleaned thoroughly each week. The filter on the control box should be vacuumed. Any build-up in the racks or rails should be blown out. The filter or filters on the vacuum hold-down systems should be removed and cleaned. All oil levels (if applicable) in the vacuum hold-down system should be checked and filled if needed. On MG Series routing systems, the Y-axis Ball Screw should also be lubricated with oil or lithium grease. MT Series routing systems will need to follow this procedure for the X-axis Ball Screw as well.

Some older routing systems use a Lead Screw on the X- and Y-axes. These systems should be lubricated with a non-aerosol, silicon-based lubricant. Using anything other than what is recommended may lead to premature wear of the Lead Screw.

Bi-Monthly

Operators should grease all the linear bearing cars at least twice a month for the ***MultiCam*** Router. Jogging the machine while applying slight pressure in the opposite direction will help force the grease into the bearing cars and incorporate the grease throughout all of the bearings.

Monthly

The racks, Y- and Z-axes screws, and X-axis screws on the MT Series routing system should be cleaned with a scrub brush and degreaser once a month following the same lubrication procedure described earlier in addition to the daily and weekly cleaning for all ***MultiCam*** Routers. Once the racks are clean, operators should apply a bead of the lithium-based grease.

Quarterly

The X- and Y-axes covers should be removed from the ***MultiCam*** Router every three months so that operators can inspect the transmission or gearboxes for wear. Operators can engage the pinion all the way into the rack by loosening the mounting bolts of the transmission or gearbox and pushing the assembly up. The tabletop bolts should be checked to ensure they are all tight.

Bi-Annually

The electrical cabinet should be checked during the bi-annual maintenance performed on the ***MultiCam*** Router. Operators should turn off the power before opening the cabinet and then vacuum out any debris that may have entered the cabinet during regular working conditions. Operators should not use compressed air when cleaning out the cabinet as this action may cause unwanted particles to short out some of the electrical components.

MultiCam recommends that the belts on all transmissions and Lead Screw Z-nuts be replaced every other year.

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Maintenance Log

Date	*Support Personnel*	*Maintenance Performed / Recommendations*

Fast Starts

Computerized Routing System Warranty

MultiCam, LP hereby warranties the electronics, motion controller, and motors of the Computerized Routing System, when properly used per the original design function, to be free of defects for a period of one (1) year from the date of shipment from ***MultiCam***. This warranty is provided to assure the initial purchaser that the machine and control units are free from defects in workmanship and materials. No responsibility for loss of machine time or work piece is expressed or implied. This warranty is valid to the original owner while in compliance with normal usage and the recommended maintenance as prescribed by the manufacturer. Failure to perform regularly scheduled maintenance and maintain the maintenance log will void this warranty.

If the electronics, motion controller, or motors of the Routing System are found to be defective during the applicable warranty period in accordance with the following specific procedure, ***MultiCam*** will, at its option, either repair or replace the parts at no charge. Replacement parts may be refurbished. All parts that are replaced shall become the property of ***MultiCam***. This warranty does not include the following items:

vacuum filters	timing belts	hoses
dust brushes	tooling	items consumed in normal use

Also excluded from the warranty is any damage to the Routing System resulting from, but not limited to, shipping, accident, disaster, misuse, abuse, or any unauthorized modification of the product

MultiCam makes no other warranty or representation of any kind, express or implied, and all implied warranties of merchantability and fitness for a particular purpose with respect to the equipment are disclaimed.

Specific Warranties

Spindle Motors

MultiCam will warranty its spindle motors for a period of 3 months from the time of installation. At any time, misuse, abuse, or use of the spindle motor outside of the manufacturer's operating specifications will not be accepted as defective and will void the warranty. After the initial 3 months, ***MultiCam*** will warrant the spindle motor to be free from defects for a period of 9 months. Spindle bearings are not covered during this 9-month period.

Vacuum Pumps

Positive Displacement systems provided by ***MultiCam*** have a standard 12-month warranty, which allows for the repair or replacement of parts at no charge. Replacement parts may be refurbished. All parts that are replaced shall become the property of ***MultiCam***. Exclusions match those listed above.

Reitschle high-pressure vacuum pumps carry a 12-month warranty covering manufacturing defects.

Limited Warranty Procedures

Inside the USA

Customers must first contact their local authorized ***MultiCam*** Technology Center to issue a purchase order, which is required for any parts that must be returned to ***MultiCam***. When replacement parts are shipped, an RMA number/tag will be included and must be prominently displayed on the outside of the return package as well as on the accompanying packing slip. All parts must be returned within 21 days, or the invoice for the parts will be due. The customer assumes cost of shipment for returned items. ***MultiCam*** will not issue Call Tags for any machine part or equipment.

Outside the USA

Please contact the local ***MultiCam*** Distributor for specific instructions.

Fast Starts

Warranty Registration List

The following documents must be returned to ***MultiCam*** in order for the warranty to be registered. Some documents may not apply to every ***MultiCam*** customer.

* Colombo Spindle Warranty Card
* EuroSpindle Warranty Card
* High Pressure Vacuum Warranty Card
* Positive Displacement Warranty Card
* Multicam Routing System Warranty Card

These documents should be faxed or mailed to ***MultiCam*** within 30 days of the machine delivery and installation, with the machine model and serial number, as well as with the company information, included in the correspondence.

Fax Number: (972) 929-4071

Mailing Address:

MultiCam
Attn: Warranty Registration
PO Box 612048
Dallas, TX 75261

Fast Starts

Innovation. Quality. Performance. Best in Industry. Worldwide.

Keypad and Functions

top

Handheld Keypad 3
Spindle Warmup 4
Emergency Stop (E-Stop) 5
Hot Keys for the Router 5
- Go Home 6
- Feedrate Override 7
- Jog Z-axis 7
- Spindle RPM Override 7
- Pause 8
- Set Surface 9
- Start 10
- Set Home 10
- Move 11
- Jog X- and Y-axes 12
- Spindle Test 13
- Menu 13
- Set Max Depth 14
- Shift 14
- DNC 15
- Bar Code 16
- Drive Enable 17
- Cancel 17
- Jog Speed 17
- Help 18
- Soft Homes 18
- Enter 19
- Find Home 19
- Park Z Up or Down 19
- Spindle RPM 20
- Fasmer Surface 21
- Spindle Mode 22
- Proximity Restart 23
- Return to Home 0 26
- Park X High or Low 27
- Mister Test 27
- Adjust Max Depth 28
- Park Y High or Low 28
- Open Chuck 29
- Drive Disable 29
- Controller Information 30
- Execute Self Test 30
- Set Soft Homes 31

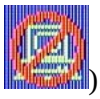
Advanced 32

Advanced Features 34
Dry Run 34
Rotate File 35
Step and Repeat 36
Bar Code Scanner 38
Safety Mat 38
Dual Start 39

Handheld Keypad

The ***MultiCam*** graphical keypad provides more flexibility in accessing the Router by allowing the operator to be closer to the work surface when setting up jobs. With the mobility of this unit, operators experience fewer problems pressing wrong keys by maintaining visual contact with the keypad. The display also allows for the standard 4 lines of text at the top of the screen to indicate the machine status as well as a graphical depiction of the cut file.

Shift Mode is indicated by the arrow at the bottom right (i.e., Normal Mode , Shift Mode).

The computer connection is also indicated at the bottom right (i.e., connected , not connected ).

Operators may press the X-axis Down Jog Arrow to view additional menu items in the various menus available on the keypad.

The keypad also provides direction with the color scheme: Blue keys indicate adjustments and settings; Green keys indicate action or movement; and the Yellow key indicates caution or pause. The Emergency Stop (E-Stop) red button located at the top of the keypad allows the operator to immediately shut down or discontinue the current job.

Keypad

Spindle Warmup

The Spindle Warmup is a procedure required by spindle manufacturers before applying load so that the bearings, bearing housings, and shaft are allowed to expand to their design dimensions. Applying a load to a cold spindle will load the bearing unevenly, causing bearing failure. Operators should insert a tool into the spindle prior to warmup. If the spindle is air or water cooled, then the operator should make sure that the cooling is turned off during spindle warmup.

After booting the machine and Finding Home, the operator will receive the following prompt from the keypad:

The default tool is #1, so the operator can press Enter to warm up the spindle with this tool. If another tool is preferred, the operator can press the number of that tool or press Cancel to skip the spindle warmup process.

When Enter is pressed, the machine picks up the tool and performs the spindle warmup procedure. This procedure will run at 6000 RPM for 5 minutes, 12000 RPM for 5 minutes, and then 18000 RPM for 5 minutes.

While the procedure runs, the system displays the remaining time for each RPM. Keypad

Innovation. Quality. Performance. Best in Industry. Worldwide.

Emergency Stop (E-Stop)

Pressing Emergency Stop (E-Stop) on either the handheld keypad or at the side of the gantry cuts power to everything except the control board of the unit, which will go into Emergency Stop mode and cut power to the inverter. This causes the spindle to coast to a stop, a process that may take several minutes. All job setup information is erased from the system, and operators should wait at least 2 minutes before engaging the machine and applying power to the motors. The E-Stop button must be pulled out when the main power switch is turned on.

Keypad

Hot Keys for the Router

Hot Keys are the keys on the ***MultiCam*** Handheld Keypad which access functions without using the menu system. The operator can access different aspects of the system using the Hot Keys, which are listed under Normal Hot Key Functions and Shifted Hot Key Functions.

Normal Hot Key Functions *(as shown on keypad)*

Go Home	Feedrate Increase	Jog Z Up	Spindle RPM Increase	Pause
Set Surface	Feedrate Decrease	Jog Z Down	Spindle RPM Decrease	Start
Set Home	Move (7)	Jog Up (8)	Spindle Test (9)	Menu
Set Max Depth	Jog Left (4)	5	Jog Right (6)	Shift
DNC	Bar Code (1)	Jog Down (2)	Drive Enable (3)	Cancel (Exit)
Jog Speed	Help (+/-)	0	Soft Homes (.)	Enter

Additional hot key features can be accessed when the keypad buttons are pressed in combination with Shift .

Shifted Hot Key Functions

Find Home		Park Z Up	Spindle RPM	
Fasmer Surface		Park Z Down	Spindle Mode	Proximity Restart
Return to Home 0,0		Park X High	Mister Test	
Adjust Max Depth	Park Y High	Open Chuck	Park Y Low	
		Park X Low	Drive Disable	
	Controller Info	Execute Self Test	Set Soft Homes	

Keypad

Go Home

The Go Home function is a return to the current 0,0 Soft Home and Tool Lift. The coordinates can be modified, but most operators prefer using soft homes for establishing different home coordinates during the cutting process.

Press Go Home .

Hot Keys Keypad

Feedrate Override

The Feedrate Override function allows the operator to adjust the X,Y travel of the machine and to change speeds from as little as 10% to as much as 100%. This affects the feedrate or cut speed when the cutter is moving through the material. The Feedrate Override has no effect when the machine is executing a rapid move above the material.

The cut speed will be shown on the keypad display as a percentage of the actual speed: *Feed%= ###*. The Feedrate Override determines the percentage by which the operator adjusts the cut speed.

1. Press Feedrate Override to adjust the feedrate in 1% increments when a file is being executed on the machine.

2. Hold Feedrate Override to adjust the feedrate in faster moving 1% increments when a file is being executed on the machine. Hot Keys Keypad

Jog Z-axis

1. Press and hold Z-axis Up Jog Arrow or Z-axis Down Jog Arrow to raise or lower the spindle.

2. Press Jog Speed to increase or decrease the Jogging speed of the spindle. Hot Keys Keypad

Spindle RPM Override

The Spindle RPM Override function allows the operator to control the RPM of the spindle from minimum to maximum levels. The spindle RPM can be adjusted while a file is being executed, and the current spindle RPM will be shown on the keypad display as *RPM=####*. The spindle speed will be set to the new value whenever the system receives a spindle speed command either automatically while a file is being executed or manually when the operator uses the Hot Keys.

1. Press Spindle RPM Increase or Spindle RPM Decrease to adjust the spindle RPM by 100 RPM increments.

2. Hold Spindle RPM Increase or Spindle RPM Decrease to adjust the spindle RPM by faster moving 100 RPM increments. Hot Keys Keypad

Pause

The Pause function allows the operator to instantly stop any motion of the machine or any file. The Z-axis will move to the Tool Lift position, and the spindle will shut off. At this point, the operator can change any 2D cutting parameters or Jog the spindle out of the way to get a better look at the cut. When *Continue* is selected, the machine will automatically return to the position before Pause was activated and continue with the commanded motion or the file.

1. Press Pause to stop the machine for any reason.

2. Press Jog Arrows until the blinking cursor is over the first letter of the command to be executed.
   a. **Continue** finishes out the operation or file originally executed.
   b. **Cancel** ends the operation or file executed and shows *Motion Cancelled...* on the keypad display.
   c. **Jog** allows the operator to Jog the machine to any location within the table parameters.
   d. **Params_2D** allows the operator to make any quick changes in the 2D cut parameters.

3. Press Enter to accept the command.

Hot Keys Keypad

Set Surface (Z=0)

The Set Surface function allows the operator to identify and set the surface of the material or the software Z reference point as the Z-axis Home (Z=0). Once set, the surface will not have to be reset unless the system is shut off or the material thickness changes.

1. Make sure the vacuum pump is on. The vacuum pump will suck the material down so that the router can read the surface dimensions properly.

2. Press Jog Arrows to position the tool bit anywhere over the material.

3. Place the grounding collar on the ATC mounting groove in the spindle tool holder.

4. Press Set Surface .

5. Place the surface block on top of the material underneath the tool.

6. Press Z-axis Jog Arrows to lower the spindle until the bit is just above the surface block. The operator should monitor the Jog speed when lowering the Z-axis as the machine will only recognize the surface block when 0 is pressed. The spindle could crash into the material or the surface block and break the bit.

7. Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

8. Remove the grounding collar and surface block before beginning work.

Hot Keys Keypad

Start

Pressing Start will begin a job file or execute a Cut Utility. When a file has been completed, the operator can press Start to activate the replay buffer and begin the file again. Operators should never leave the machine unattended during the cutting sequence.

Hot Keys Keypad

Set Home (0,0)

The Set Home function allows the operator to identify the exact location of X=0, Y=0 on the material to be cut. This is the Soft Home location 0. The file that is to be executed on the table will use this location as the software reference point home or origin (0,0): the machine will begin and return to this new origin, and the X and Y coordinate locations are stored in the controller's memory. The location of the Set Home position will be the new origin (0,0) until a new home is set or the current home is cleared by selecting Clr_Home from the Utility sub-menu. The software reference point **must be** the material home point.

1. Press Jog Arrows to place the **center of the bit** over the front right corner of the material.

2. Press Set Home .

3. Press Enter to accept. This will store the absolute X and Y coordinate locations in the controller memory.

Hot Keys Keypad

Move

The Move function allows the operator to enter exact values for X, Y, Z (0,0,0) where the first entry is X (length), the second entry is Y (width), and the third entry is Z (height). The X, Y, and Z values will reference from the Hard Home position until the Set Home and Set Surface functions have been completed. At that point, the X- and Y-axes will reference from the Set Home position. The Z value will reference from the surface of the material.

1. Press Move

.

2. Enter the desired value for X using the number keypad (e.g., 4.0) and press Enter .

3. Enter the desired value for Y using the number keypad (e.g., 4.0) and press Enter .

4. Enter the desired value for Z using the number keypad (e.g., 0.0) and press Enter .

5. Press Enter when the "?" is displayed to accept all settings and move the machine to the new coordinates.

6. Press Cancel to return to the main display.

The Move function can also be used to verify a specific location.

1. Press Move

.

2. Press Enter three times to verify the current location of the X-, Y-, and Z-axes.

   The operator should press Enter to keep the axis value from changing when that value is shown on the screen.

Hot Keys Keypad

Jog X- and Y-axes

The Jog Arrows

are used to Jog the location of the spindle and to move through the menu.

Jog Speed can be pressed to change the Jogging speed of the machine.

1. Left Arrow

is the Y-axis Positive Jog Key and Left Scroll Key in the menu.

2. Up Arrow

is the X-axis Positive Jog Key and Line Up Key in the menu.

3. Down Arrow

is the X-axis Negative Jog Key and Line Down Key in the menu.

4. Right Arrow

is the Y-axis Negative Jog Key and the Right Scroll Key in the menu.

Hot Keys Keypad

Spindle Test

The Spindle Test function tests the controller output to the spindle motor, which will stay active until the test key is pressed again or the test is cancelled.

1. Press Spindle Test

.

2. Press Spindle Test

again to run the spindle test or press Enter to run Spindle Warmup.

3. Press Cancel to stop the Spindle Test.

Hot Keys Keypad

Menu

Menu items help the operator handle many of the functions of the ***MultiCam*** CNC Cutter by providing the ability to set feedrates and other parameters, run tests, and cut basic shapes. With the addition of some advanced features of the ***MultiCam*** Router, the operator can also adjust tools or duplicate shapes which create G-Code files quickly.

Many of the Menu Items are standard for all ***MultiCam*** Routers but with different options, such as Automatic Tool Change and MultiHead. Some options have further additional items.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

to move the cursor to the specific menu item. Certain menu items will then pull up a sub-menu with more selection. Not all menu items will be applicable for some systems.

Hot Keys Keypad

Set Max Depth

The Set Max Depth function allows the operator to set a cutting limit so that the tool will not plunge below that setting. The ***MultiCam*** Controller has a maximum depth safety feature that functions for all file types and modes and will truncate any depth greater than the maximum depth. Setting the maximum depth is similar to setting the surface, and the maximum depth also becomes the cut depth. The operator may set a shallower cut depth after the maximum depth has been configured.

1. Press Jog Arrows to position the tool bit anywhere over the tabletop or at the location of the required maximum depth.

2. Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

3. Press Set Max Depth to access the maximum depth for the bit.

4. Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

5. Press Z-axis Jog Arrows to lower the spindle until the end of the tool is just above the surface block.

6. Press and hold Zero to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

7. Remove the grounding collar and surface block before beginning work. Hot Keys Keypad

Shift

Pressing Shift before pressing any Hot Key will allow the operator to toggle between *Normal* Hot Key and *Shifted* Hot Key functions.

When Shift has been pressed, the Shift LED will illuminate to indicate that the keypad is now in *Shifted Mode*. Hot Keys Keypad

DNC

The Distributed Numerical Control (DNC) function allows the operator to access files directly from the hard disk of the host PC by using the keypad at the machine. This allows the computer and the machine to be in different locations, which frees up the operator to view the progress of the cutting sequence.

1. Make sure that JobServer or JobNameServer has been activated on the computer before selecting this function.

2. Press DNC .

   The keypad will display a list of the directories in the DNC path after the system has validated and verified.

3. Press Jog Arrows until the blinking cursor is over the directory to be entered.

4. Press Enter . The keypad will display the DNC files that are available in that directory, and directories or folders and any associated subfolders are displayed between brackets [ ]. Press Enter while the blinking cursor is located over the 2 decimal points at the beginning of the directory to return to the list of directories if necessary.

5. Press Jog Arrows until the blinking cursor is over the first letter of the file to be executed. Z-axis Jog Arrows will page up or down one screen at a time, while X-axis Jog Arrows will move up or down one line at a time.

6. Press Start .

   If the operator just presses Enter over the blinking cursor, he or she will only receive a display of the file name, size, and date/time created.

7. Press one of the following Hot Keys to execute the required command or press X-axis Down Jog Arrow once to access the Dry Run function and once more to access the Step and Repeat function.

   a. Start will execute the selected job file.

   b. Cancel will end the selected job file.

   c. Pause will allow the operator to change a tool, set up a 2D parameter, or set surface or max depth while holding the file in queue.

   Once the file has completed running, the keypad display will show how long the selected file ran and will automatically return to the directory. Hot Keys Keypad

Bar Code

The Bar Code function allows the operator to scan a label with the Bar Code Scanner, which will automatically call up and download the correct job for execution on the machine. In addition, ***MultiCam*** provides JobPreviewer to print bar code labels and tool path previews with operator notes. The Bar Code Scanner is limited to 48 characters excluding the initial DNC path that is set up in JobPreviewer, JobServer or JobNameServer (e.g., C:\DNCfiles\).

1. Press Bar Code to execute the Bar Code function. This will set the display to show when the system is bar code ready.

2. Depress the trigger on the bar code scanner and scan the bar code label printed on the JobPreviewer sheet.

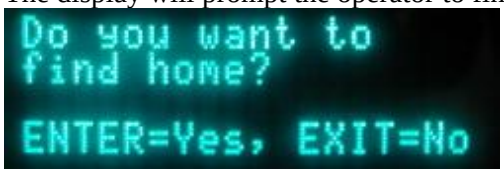
   Example Only:

3. Press Start when prompted to begin the job. Hot Keys Keypad

Drive Enable

1. Press Drive Enable to enable all Servo Drives after a system error.

The display will prompt the operator to find home after re-enabling the motor drives.



2. Press Enter to ensure proper machine operation. Hot Keys Keypad

Cancel

Press Cancel to cancel or exit any selected function or menu item. In some cases, this needs to be pressed twice to make sure the function has been completely canceled. Hot Keys Keypad

Jog Speed

Press Jog Speed to toggle between 3 different speeds at which the machine can Jog the ***MultiCam*** Router. The keypad display will prompt the operator with *Jog= FAST*, *MED*, or *SLOW*.

Hot Keys Keypad

Help

The Help function allows the operator to learn the functions of any Hot Key selected. There are 2 modes of the Help function: *Normal* and *Shifted*. In *Normal* mode, the operator can review the standard function of any Hot Key on the display. In *Shifted* mode, the operator can review the second or shifted function of that key on the display.

1. Press Help to access the Hot Key Help Feature.

2. Press any key for a brief description of that key in *Normal* mode.

3. Press Shift and any key for a brief description of that key in *Shifted* mode.

4. Press Cancel twice to exit help. Hot Keys Keypad

Soft Homes (.)

The Soft Homes function allows the operator to have multiple fixture setups to be quickly accessed during the cutting procedure. The Soft Homes Hot Key also allows the operator to input a decimal into a numeric value.

1. Press Soft Homes .

2. Enter the appropriate number (1-9) to assign the proper value for the new home position (e.g., 6).

3. Press Enter to accept the Soft Home. The display will acknowledge that the particular home position has been selected.

4. Press Shift and Soft Homes to set new Soft Home locations. Hot Keys Keypad

Enter

Press Enter to accept functions, settings, and menu items as directed.

The display may provide a question mark "?" when processing certain functions. The operator can press Enter to verify that the action should take place. Hot Keys Keypad

Find Home

The Find Home function allows the operator to return each axis to the machine home position in order to relocate the homing targets. The Spindle Head Assembly will move to the right front of the table and index all X, Y, Z components starting with the Z-axis.

1. Press Shift and Go Home to index the gantry, carriage, and Z assembly off of the limit switches after one of the following actions have been completed:
   a. The machine has just been turned on.
   b. The machine has stalled or lost its position coordinates.
   c. The machine has been stopped because the operator has pushed E-Stop.

2. Refer to the display as it will indicate when the system locates the targets for each axis. The spindle drive will be enabled when the Find Home function has been activated. The operator should take special care ATC systems in making sure the air is **on** and the dust collector is **open**. Hot Keys Keypad

Park Z Up or Down

The Park Z function will move the spindle head assembly to either the absolute Z=0 position or all the way up in a process which allows for tool changes including easily moving the pressure foot.

1. Press Shift and Z-axis Up Jog Arrow to raise the spindle head assembly to the absolute Z=0 position.

2. Press Shift and Z-axis Down Jog Arrow to lower the spindle head assembly to the Tool Lift position.

Hot Keys Keypad

Spindle RPM

The Spindle RPM refers to the revolutions per minute that the spindle will spin, and the spindle RPMs may vary depending on the type of material being cut. Operators can enter the desired spindle RPM for manual operations such as Material Cutoff or Table Mill. The Spindle RPM can also be set in the cut file and will override what has been entered at the keypad. This value can be set from the keypad or in the Params_2D sub-menu. The keypad display is shown below.

1. Press Shift and Spindle RPM Increase .

2. Enter the desired spindle RPM value using the number keypad (e.g., 18000).

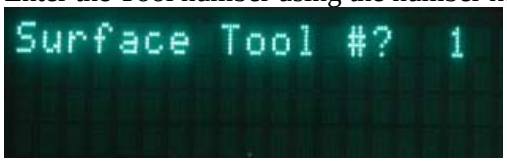
3. Press Enter to accept the value. The display will show that the speed has been set at the value entered and will then default to the main screen.

Hot Keys Keypad

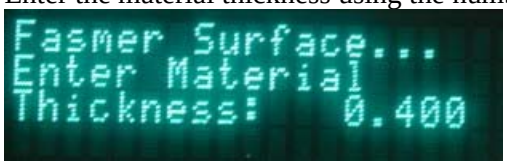
Fasmer Surface

The Fasmer Surface function is useful when working with material that has an irregular surface.

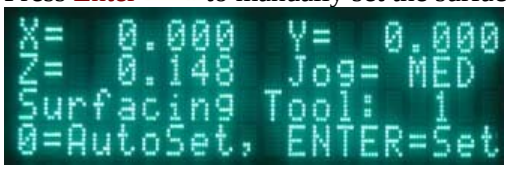
1. Press Shift and Set Surface to access the Fasmer Surface function.
2. Enter the Tool number using the number keypad (e.g., 1) and press Enter.



3. Enter the material thickness using the number keypad (e.g., 0.400) and press Enter.



4. Place the surface block on the spoilboard.
5. Press Jog Arrows to position the tool bit over the surface block.
6. Press Enter to manually set the surface.



7. Press Z-axis Jog Arrows to lower the spindle until the bit is just above the surface block. The operator should monitor the Jog speed when lowering the Z-axis as the machine will only recognize the surface block when Zero is pressed. The spindle could crash into the material or the surface block and break the bit.
8. Press and hold Zero to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Hot Keys Keypad

Spindle Mode

The Spindle Mode function allows the operator to enable or disable a tool. Changing a tool can be accomplished without the spindle inadvertently being turned on. Once the spindle has been disabled, the operator cannot activate the spindle from the keypad or a job file.

1. Press Shift and Spindle RPM Decrease to enter Spindle Mode.

2. Enter the Tool # for No Spin using the number keypad. The operator may enter Zero to disable the function.
3. Press Enter to accept the selection and return to the main display.

Hot Keys Keypad

Proximity Restart

The Proximity (Prox) Restart function is an advanced feature that allows the operator to begin a file by locating the end position based on the original Home location. The controller will search through the file to locate the nearest point and base the coordinates of the restart on the Home position from where the file was originally executed.

Proximity Restart is used any time a file is interrupted and restarting the entire file is not practical. The operator can return to the original Home position to restart the program as the configuration is based on where the cutter stopped in relation to where the cutter originally began. This command will function through the internal replay buffer of the machine. The following directions describe how to operate Prox Restart for older systems:

1. Press Shift and Start from the Home position where the file was originally executed.

2. Press Jog Arrows to move the spindle to a position within the restart tolerance of the desired point for restart.

3. Press Enter to begin the search.

   The keypad display will indicate the coordinates when a point within the restart tolerance is found. If no points are located within the restart tolerance, the system will default to the main display. At this point, the operator should repeat the Proximity Restart process.

4. Press Start to restart the file at the current location or press the X-axis Up Jog Arrow to access the closest point in the file. Spindles can begin at the very beginning and retrace steps.
   a. If the point is located over a Tool UP Move, follow the keypad display and repeat step 3.
   b. If there is a power failure, the operator will need to re-index the machine and reset Surface and Max Depth.

For systems configured with Suite4.1 and Router Inits v.8.11.14, Prox Restart has been further modified. The following directions describe how to access the new Prox Restart function, select a particular mode, and complete basic troubleshooting.

Accessing Prox Restart

1. Jog to the appropriate location to begin the Prox Restart.

2. Press Shift and Start to access Prox Restart.

   - If the job is a single sheet, then skip to Step #5.
   - If the job is not a single sheet, then continue with Step #3.

3. Press Spindle RPM Override to select the sheet.

4. Press Start at the beginning of the selected sheet.

5. Press Enter to search for the X,Y location. The machine will look for a point within a radius of 0.300” (7.620mm).
   - If the machine cannot find the point, then skip to **Troubleshooting Prox Restart**.
   - If the machine can find the point, then select a mode in which to begin the cutting sequence. Operators may access the different modes through either specific hot keys or keypad selection (shown below).

Operating in Hot Key Mode

The following hot keys have the same functionality as the same hot keys in Suite4 JobConsole. The system looks for a point closest to the current location based on the hot key pressed, and these hot keys operate independently of the mode currently visible on the display.

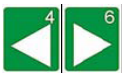
- The X-axis Jog Arrows allow the technician to locate the previous or following point (e.g., perform the next Distance command: X-axis Up = Forward Distance, X-axis Down = Backward Distance).

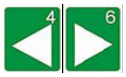
- The Z-axis Up and Down Jog Arrows allow the technician to locate the previous or next PD (i.e., Z-axis Up = Previous PD, Z-axis Down = Next PD).

- The Feedrate Override Arrows allow the technician to locate where the previous or next tool will begin cutting (i.e., Feedrate Override Up = Previous Tool, Feedrate Override Down = Next Tool).

Operating in Keypad Selection Mode

Additional modes can also be accessed, which can be beneficial for operators who cannot find the starting point in the file through the above modes. Operators may select a Proximity Restart Mode through the keypad by completing the following:

1. Press Help and the Y-axis Jog Arrows  to enter the Prox Restart keypad selection mode.

2. Press Help to cycle through the Prox Restart modes and Y-axis Jog Arrows  to navigate forward and backward within the selected mode. Each mode is described below.

    - Dist – Move forward and backward based on a distance. When in Distance Mode, technicians can press Jog Speed to cycle through 3 distances (i.e., 0.010”, 0.050”, 0.100” [1mm, 2mm, 3mm]).
    - PD – Move to the previous and next pen down (PD) commands in the job file (e.g., M11, M12)
    - PU – Move to the previous and next pen up (PU) commands in the job file (e.g., M21, M22)
    - Tool – Move to the previous and next tools in the job file
    - Mcmd – Move to the previous and next source line or Motion Command in the job file

Troubleshooting Prox Restart

An error message of *Location Not Found* will appear if JobConsole cannot find a matching Marker Type based on the mode selected above. When this happens, the system will automatically re-enter Prox Restart at the current location, and the technician will need to select a different mode to continue the Prox Restart.

For example, if a technician uses Mode PD to move to the next PD in the job file and then switches back to Mode Dist, he or she cannot back up in Dist mode. Since JobConsole is at the beginning of the contour, it will not be able to back up and therefore will report the error message *Location Not Found*.

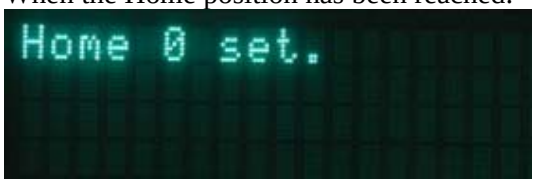
Hot Keys Keypad

Return to Home 0

The Return to Home 0 function will return the spindle to the last recorded set home position or Soft Home location 0. If power has been lost or the machine has been rebooted, Home 0 will become the new Hard Home.

Press Shift and Set Home .

When the Home position has been reached:



Hot Keys Keypad

Park X High or Low

The Park X function will move the spindle and gantry assembly to a set park position that could be as far away as the maximum length of the table or all the way back in X-axis. This allows the operator to reload material on the table or examine work without interference from the spindle.

1. Press Shift and X-axis Up Jog Arrow to move the spindle and gantry assembly to the high or farthest set park position.

2. Press Shift and X-axis Down Jog Arrow to move the spindle and gantry assembly to the low or closest set park position.

Hot Keys Keypad

Mister Test

The Mister Test function will test the controller output to the Automatic Misting System. On systems with no Automatic Misting System, this command will do nothing. This option can also be used to prime a misting unit if applicable.

1. Press Shift and Spindle Test to activate the Mister Test.

2. Press Cancel to stop the Mister Test and return to the main display.

Hot Keys Keypad

Adjust Maximum Depth

The Adjust Maximum Depth function may be necessary when the cutter travels shallower than expected, usually due to the variance in material thickness. Operators can adjust the maximum depth without having to reconfigure all settings.

1. Press Shift and Set Max Depth.

2. Press Cancel to clear the maximum depth without having to reboot the machine or press Z-axis Jog Arrows to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.
3. Press Enter to accept the setting.

Hot Keys Keypad

Park Y High or Low

The Park Y function will move the spindle assembly the maximum length of the gantry to the right or left in Y-axis. This allows the operator to examine the work without interference from the spindle.

1. Press Shift and Y-axis Left Jog Arrow to move the spindle assembly to the left side of the gantry.

2. Press Shift and Y-axis Right Jog Arrow to move the spindle assembly to the right side of the gantry.

Hot Keys Keypad

Open Chuck

The Open Chuck function allows operators to verify whether the chuck is open or closed. Once tools have been removed or modified in the ATC, the chuck must be closed before work can be continued.

1. Press Shift and Five to check the status of the chuck.

2. Press Eight to open the chuck if the display shows the chuck is closed. Tools can be removed or modified from the ATC.

3. Press Shift and Five to access the chuck status function once tools have been configured.

4. Press Two to close the chuck or Cancel to exit the Open Chuck screen.

Hot Keys Keypad

Drive Disable

The X-, Y-, and Z-axis drives will be disabled in the event of a system failure or a mechanical obstruction to prevent possible damage to the individual drives. The operator also has the ability to disable the drives from the keypad. Once the drives have been disabled, the machine cannot be operated by using the keypad.

Press Shift and Drive Enable to disable all drives.

When the drives associated with servo systems are disabled, the power to the motors is also disabled. The motors associated with stepper systems still have power after the drives are disabled, but there is no output from the control board.

Hot Keys Keypad

Controller Information

The Controller Information function provides information on the versions of software that are loaded in the controller.

1. Press Shift and Help to display the controller information. This includes the version of machine Init files and firmware, as well as the types of Interface and Control boards.

2. Press X-axis Down Jog Arrow to access Ethernet information while viewing the controller information.

3. Press DNC to change the IP address if necessary.

Hot Keys Keypad

Execute Self Test

Press Shift and Zero to access the self test, which runs a diagnostic on the following:

- Memory verification
- Rectangle routing
- Surface block checking
- Automatic Tool Changer (ATC) testing
- Origin (Home) testing

Hot Keys Keypad

Set Soft Homes

The Set Soft Homes function allows the operator to establish 9 programmable Soft Home positions to record multiple home positions that can be used in supporting multiple cutting fixtures. These homes stay recorded in memory even after power has been removed from the system.

1. Press Jog Arrows to position the **center of the bit** where the new Soft Home location should be set.
2. Press Jog Speed to change the Jogging speed if necessary.
3. Press Shift and Soft Home .

4. Enter a number between 1 and 9 using the number keypad. If the operator wants to save the Z value in the Soft Home, he or she should contact a ***MultiCam*** technician for assistance.
5. Press Enter to record the new Soft Home.

Hot Keys Keypad

Advanced

Advanced Hot Keys are accessed by pressing Shift and Help before pressing any of the keypad numbers 1-9. The operator will be in the main information screen upon pressing the Shift and Help sequence the first time.

Main Information Screen

Line 1: Init File Versions
Line 2: Firmware Version
Line 3: Controller and Interface Board rev and ID
Line 4: Keypad version

Ethernet Information Screen

Line 1: “Ethernet Information”
Line 2: IP Address
Line 3: Net Mask
Line 4: DHCP Server Address (Only shows up if DHCP enabled)

Press DNC from this screen to be prompted to enable DHCP or specify an IP address.

Power Information Screen

Line 1: “Power Information”
Line 2: Incoming 24V Power to the Board
Line 3: “Recorded Power”
Line 4: “Glitches:” <number of glitches> If this number exceeds 6, a job will be stopped with a power failure.

MODBUS Information Screen

Line 1: “MODBUS Information”
Line 2: Packets sent
Line 3: Number of Timeouts Occurred (TMOs) and Packets Blocked

Drive Temperature Information Screen

The drive temperature information is only applicable to HP4 systems.
Line 1: “Drive Info” Peak Temperature
Line 2: X Drive Temperature, Y Drive Temperature
Line 3: Z Drive Temperature, Xb Drive Temperature
Line 4: Bus Voltage

Inputs Information Screen

Each column shows the status of the specific limit.

- 0 = off
- 1 = on

Machine Information Screen

Line 1: “Machine Information”
Line 2: Model and Type
Line 3: Serial Number

Keypad

Advanced Features

Advanced Features help the operator to be more productive by previewing a file before cutting the material with Dry Run, rotating a file to match the angle of the sheet with Rotate File, cutting an entire sheet of shapes using the Step and Repeat function, accessing the bar-coded information with the Bar Code Scanner, and with using the Safety Mat as an option for pausing the cutting sequence. Some routers are also available with Dual Start, an option to maximize the cutting time by creating 2 separate work areas on the same machine.

Keypad

Dry Run

The Dry Run function is an advanced feature that allows the operator to preview the file to be cut while leaving the feedrate override active. The spindle will stay in the Tool UP position, never entering the material. The keypad displays *Tool 1 Up* when the cutter would normally be in the UP position moving between contours. The keypad displays *Tool 1 Down* when the cutter would normally be in the material cutting the contours. This command will function through the DNC or through the internal replay buffer of the machine.

1. Make sure that JobServer or JobNameServer has been activated on the computer.

2. Press DNC . The keypad display will show a list of the directories in the DNC path.

3. Press Jog Arrows until the blinking cursor is over the directory to be entered.

4. Press Enter . The keypad will display the DNC files that are available in that directory, and directories or folders and any associated subfolders are displayed between brackets [ ]. Press Enter while the blinking cursor is located over the 2 decimal points at the beginning of the directory to return to the list of directories if necessary.

5. Press Jog Arrows until the blinking cursor is over the first letter of the file to be executed.

6. Press Start .

7. Press X-axis Down Jog Arrow once.

8. Press Enter to start the Dry Run process. The operator can press Cancel to abort the selected job file or Pause to change a tool, set up a 2D parameter, or set surface or max depth while holding the file in queue.

Advanced Keypad

Rotate File

The Rotate File function is used to rotate the program to match the material. Operators may find it impractical to square material sufficiently when heavy materials are loaded onto the table, so this function allows the machine to adjust the parameters of the job file.

1. Press Jog Arrows to position the **center of the bit** over the rear right corner of the material and press Set Home .

2. Press the X-axis Down Jog Arrow to set the Rotation Point. The keypad will briefly display the rotation setting and then switch back to the main display.

3. Press Start to begin the job file. The operator will be prompted to accept or reject the rotation.

Advanced Keypad

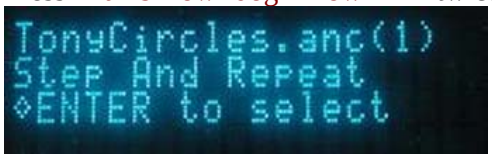
Step and Repeat

The Step and Repeat function is used to make a single job file cut multiple times. Operators can program a single part of the cutting sequence and include the size of the material to be cut without having to create a larger file with the detailed parts and motions.

1. Make sure that JobServer or JobNameServer has been activated on the computer.
2. Access the file by pressing DNC .
3. Press Jog Arrows until the blinking cursor is over the directory to be entered.
4. Press Enter . The keypad will display the DNC files that are available in that directory, and directories or folders and any associated subfolders are displayed between brackets [ ]. Press Enter while the blinking cursor is located over the 2 decimal points at the beginning of the directory to return to the list of directories if necessary.
5. Press Jog Arrows until the blinking cursor is over the first letter of the file to be executed. Z-axis Jog Arrows will page up or down one screen at a time, while X-axis Jog Arrows will move up or down one line at a time.
6. Press Start to access the particular file.

If the operator just presses Enter over the blinking cursor, he or she will only receive a display of the file name, size, and date/time created.

7. Press X-axis Down Jog Arrow twice to access the Step and Repeat function.



8. Press Enter to start the Step and Repeat function.

9. Enter the sheet size in X-axis using the number keypad (e.g., 119.565) and press Enter .

10. Enter the sheet size in Y-axis using the number keypad (e.g., 48.690) and press Enter .

11. Enter the space between the parts using the number keypad (e.g., 0.500) and press Enter .

12. Press X-axis Jog Arrows

to select whether the machine should step along the X- or Y-axis.

13. Press X-axis Jog Arrows

to select Yes or No for the Serpentine option.

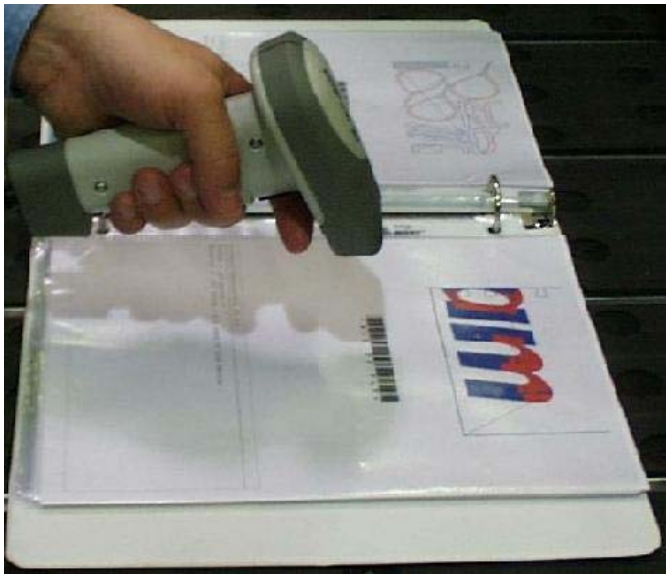
If X-axis was selected above and serpentine is selected next, then the machine will cut the parts in a line down the X-axis, move over in the Y-axis, and cut back up the X-axis. This sequence of motion increases efficiency since the machine will not make a big slew move back to the front of the machine after each row of parts.

14. Press Enter to start the cutting sequence.

Advanced Keypad

Bar Code Scanner

The ***MultiCam*** Bar Code Scanning Interface has been developed to reduce setup time, minimize errors, and work directly with shop floor work orders.The bar code scanner works in direct conjunction with the JobPreviewer and DNC. The JobPreviewer is used to produce shop floor work orders, which are automatically assigned unique bar code identifications. Once the machine operator scans the bar code, DNC will automatically call up the correct file and send it to the controller. This saves time and reduces the chance for error.



An excellent tool for communication of special instructions, the work order contains a graphical representation of the cut file, tool specifications, material descriptions, and user notes. The bar code label on the work order ensures that the correct job is accessed. Advanced Keypad

Safety Mat

The Safety Mat is a pressure-sensitive foot mat that will cause the machine to enter into Pause mode when the mat is stepped on during machine operations. This Pause mode can only be cleared by pressing the “Safety Mat Reset” switch at the operator station and then clearing the error at the keypad.

MultiCam advises operators to pause operations during any changing of tools or materials and recommends that operations be paused when any disruptive action (e.g., change of shift, emergency situation) occurs in the cutting area. All cutting operations should be stopped and the machine turned off before any maintenance issues are addressed. Advanced Keypad

Dual Start

The Dual Start function is available with most CNC systems and requires an Operating (op) station with options for Home 1 and Home 2. Operators who run the same job several times in a row may choose the Dual Start option to maximize the table space and increase productivity.

1. Make sure the Dual Start Module has been loaded on the machine.

2. Set Parameter 24 to 1.

3. Set Soft Homes **1 and 2**. The system will automatically recognize that Dual Start is being requested. Other Home values will not work.

   a. Press Jog Arrows to position the **center of the bit** where the first Soft Home location should be set.

   b. Press Shift and Soft Homes .

   c. Enter 1 for Home 1 and press Enter to record the Soft Home.

   d. Press Jog Arrows to position the **center of the bit** where the second Soft Home location should be set.

   e. Press Shift and Soft Homes .

   f. Enter 2 for Home 2 and press Enter to record the second Soft Home.

4. Press DNC .

5. Press X-axis Jog Arrows until the blinking cursor is over the appropriate job file.

6. Press Enter to access the job file.

7. Push button 1 or 2 on the op station to move the spindle to Soft Home 1 or 2.

8. Press Start to begin the file.

9. Press the other button on the op station to run the same job at the other Soft Home location after the first job file has been completed. Operators should make sure the material has been properly loaded at the other location before beginning the job file at that location.

Advanced Keypad

Menu Items

top

Params_2D	4
X,Y Feedrate	5
Cut Depth	6
Tool Lift	7
Z Plunge Feed	8
Spindle RPM	9
Spindle DIR	10
Rec_Home	11
Digitize	12
Digitize Commands	12
Manual Digitize	14
Rapid Shape	15
Using Rapid Shape	16
Rapid Shape Tips and Troubleshooting	18
Shutdown	19
Cut_Utils	20
Table Mill	21
Table Mill Tool Parameters	23
Test Cut	24
Test Cut Tool Parameters	25
Circle	26
Circle Shape Tool Parameters	27
Rectangle	28
Rectangle Shape Tool Parameters	29
Square	30
Square Shape Tool Parameters	31
Material Cutoff	32
RipCut	33
Polygon	34
Polygon Shape Tool Parameters	35
Radiused Rectangle	36
Radiused Rectangle Shape Tool Parameters	38
Utility	39
Clr_Home	40
No_Spin	40
Job_Time	41
ParkPos	42
SPA Trigg	43
Lube	44
PauseCfg	45
GDrillTest	46
Inv_Reset	47
Reboot	47
AdjSurf	48
DNC Mode	49

DustCtrl 51
MisterOpt 52
Tool_Comp 53
Vision 54
Focus Camera 55
Fiducial Diameter 56
Correlation 57
Retake Distance 58
Settling Time 59
Flip to Cut 60
Auto Focus 61
Fast Find 62
Camera Delay 63
Auto Find Setting 64
Manual Find Setting 65
Set Camera Scale 66
Set Camera Offsets 67
Move to Cam Offsets 70
ATC 71
Cal_Tool 72
Tool_Unld 73
CalGDrill 74
AdjGDrill 75
LoadTools 76
Adj_Tool 77
Get_Tool 78
LoadDrill 79
CfgGDrill 80
Knife 81
Set Home w/Knife 82
Knife Control 83
XY Feedrate 84
ZDn Feedrate 85
KnifeEnabled 86
Knife Lift 87
Knife Depth 88
Knife Dn Delay 89
Knife Up Delay 90
Closeness 91
Rotate Degree 92
Min. Radius 93
HomeKnife 94
Set Knife Offset 95
Knife Test 96
Knife Slider Test 97
Set Knife Res 98

KnifeUtil 99
- Cartridge Number 100
- Cartridge Config 101
- Osc Stroke 103
- Cartridge Offsets 104
- Move to Knife Offset 105
- Knife Osc Test 106
- Set Knife 1 Res 107
- Set Knife 2 Res 108

MatlHand 109
- Roller Setup … 110
- Material Pusher … 111
- Material Lifter … 112
- Air/Knife Sweeper 113
- PopUp Pins 114

Params_2D

The Params_2D sub-menu is used to set machine parameters specific to a routing machine. The display extends 4 rows, so the operator must Jog down to see additional menu items. The sub-menu items allow the operator to set certain specifications for cutting different materials or removing waste material.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “P” in Params_2D.

3. Press Enter to access the Params_2D sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the appropriate option. Certain menu items will then enter a sub-menu with more selections.

X ,Y Feedrate (Cut Speed)
Cut Depth
Tool Lift
Z Plunge Feed
Spindle RPM
Spindle DIR
Menu

X & Y Feedrate (Cut Speed)

The X & Y Feedrate refers to the rate, measured in inches per minute (IPM), that the tool will travel in the X, Y direction through the material while the tool is in the Pen Down (PD) position (i.e., the Cut Speed). These Speed settings will vary with the type of material used. Metal is usually cut at a slower speed to prevent possible took breakage, while plastic and wood generally have a faster cut speed to prevent material melting or burning.

The Cut Speed function can only be used when no feedrate has been programmed in the numerically controlled (NC) file. The machine will default to 60 IPM if the machine loses power or the controller is rebooted.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “P” in Params_2D.

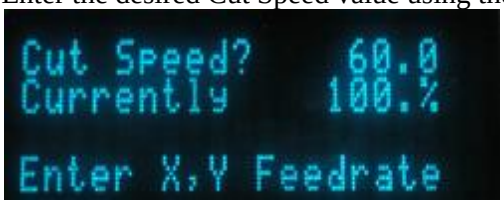
3. Press Enter to access the Params_2D sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “X” in X, Y Feedrate.

5. Press Enter to adjust the Cut Speed value.

6. Enter the desired Cut Speed value using the number keypad (e.g., 60).



The value shown next to “Currently” shows at what percentage the feedrate is set.

7. Press Enter to register the value and return to the Params_2D menu.

Params_2D Menu

Cut Depth

The Cut Depth function is used to set how deep the cutter will go below the surface on a cutting move. The Cut Depth function at the controller is only applicable for 2 dimensional HPGL files that do not have the ZD command in them, and Cut Depth has **no** effect on G-Code files or 3D HPGL files.

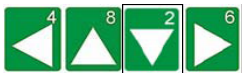
1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “P” in Params_2D.

3. Press Enter to access the Params_2D sub-menu.

4. Press Jog Arrows 

until the blinking cursor is over the “C” in Cut Depth.

5. Press Enter to adjust the Cut Depth.

6. Enter the desired Cut Depth value using the number keypad.

7. Press Enter to record the value and return to the Params_2D menu.

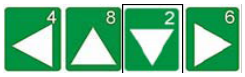
Params_2D Menu

Tool Lift

The Tool Lift function is used to raise the tool bit above the surface of the material between cuts and is only functional for controller functions such as Go Home and Material Cutoff, HPGL programs with no Tool Lift included, and G-Code programs. This lift allows the spindle to travel above and across the material without hitting the hold-down clamps. Tool Lift can be modified after the surface is set. The Go Home command will move the Z-axis to the new tool lift position.

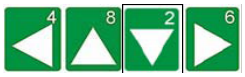
1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “P” in Params_2D.

3. Press Enter to access the Params_2D sub-menu.

4. Press Jog Arrows 

until the blinking cursor is over the “T” in Tool Lift.

5. Press Enter to adjust the Tool Lift.

6. Enter the desired Tool Lift value using the number keypad (e.g., 0.50).

7. Press Enter to record the Tool Lift value. The number entered should be a positive number, but the display will show it as a negative number since movement up from the surface is considered negative while movement down from the surface is considered positive.

8. Press Cancel to return to the main screen.

Params_2D Menu

Innovation. Quality. Performance. Best in Industry. Worldwide.

Z Plunge Feed

The Z Plunge Feed is the feedrate in inches per minute (IPM) that the tool will travel in the Z-axis direction from surface to final depth, or the speed the tool enters the material. This speed is only active for 2-dimensional HPGL files that do not have a Z Speed or Cut Speed in them. Speed settings for hard materials should be slow, while speed settings for softer materials, such as plastic and wood, should be faster.

The machine will default to 60 IPM if the machine loses power or the controller is rebooted. The Z Plunge Feed does not control the speed by which the Z-axis travels from the Tool Lift position to the surface of the material. This speed is set in the machine parameters by ***MultiCam*** to reduce total process time.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “P” in Params_2D.

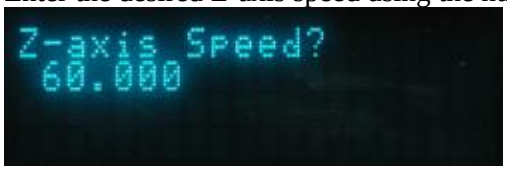
3. Press Enter to access the Params_2D sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “Z” in Z Plunge Feed.

5. Press Enter to adjust the Z Plunge Feed.

6. Enter the desired Z-axis speed using the number keypad (e.g., 60).



7. Press Enter to record the value and return to the Params_2D menu.

Params_2D Menu

Spindle RPM (Menu)

The Spindle RPM is the revolutions per minute that the spindle will spin, and spindle speeds may vary with the type of material being cut. The operator can enter the spindle speed in the desired RPMs. The spindle speed can also be set in the cut file and will overwrite what has been entered in the keypad. This value can be set from the keypad or in the Params_2D sub-menu, which is shown below.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “P” in Params_2D.

3. Press Enter to access the Params_2D sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “S” in Spindle RPM.

5. Press Enter to access the Spindle RPM function.

6. Enter the desired Spindle RPM value using the number keypad (e.g., 18000).

7. Press Enter to register the value and return to the main screen.

Params_2D Menu

Spindle DIR

The Spindle DIR allows the operator to adjust the direction the spindle will turn.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “P” in Params_2D.

3. Press Enter to access the Params_2D sub-menu.

4. Press Jog Arrows

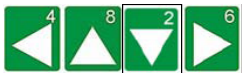
until the blinking cursor is over the “S” in Spindle DIR.

5. Press Enter to toggle between Forward (FWD) and Reverse (REV).

Params_2D Menu

Recording a Soft Home

There are 9 different recordable home positions that can be used to support multiple cutting fixtures. These homes stay recorded even after power has been removed from the system.

1. Press Jog Arrows  to position the **center of the bit** where the new Soft Home location is to be set.
2. Press Menu to access the menu system.
3. Press Jog Arrows until the blinking cursor is over the “R” in Rec_Home.
4. Press Enter to access the Recording a Soft Home function.

5. Enter the desired number (1-9) to record the new home.

Menu

Digitize

The Digitize function is used for the ***MultiCam*** Rapid Shape feature or Manual Digitizing. With either feature, the operator has the ability to duplicate simple and complex shapes on the router. This will record the outline on the host PC. Rapid Shape and Manual Digitizing both create a G-Code file that can be imported into many CAD/CAM systems. ***MultiCam*** does offer a DXF conversion file that will allow the G-Code file to be imported into many design software packages.

Before starting Digitize, the operator must verify that JobServer or JobNameServer is visible in the Windows system tray, the small box in the bottom corner by the system clock, as this means that JobServer or JobNameServer is active and running on the computer. If this icon is displayed, the operator is ready to digitize the machine. Not all ***MultiCam*** Routers carry the Digitize option, but the Digitize option can be purchased from the local distributor or from ***MultiCam***.

Menu

Digitize Commands

The Digitize Commands are utilized in Manual Digitizing and Rapid Shape, and only a few of these commands are used in Rapid Shape.

1. **Linear** (Feedrate Increase) allows straight line, point-to-point moves in these G-Code Commands: G00 for PU,G01 for PD.

2. **Spline** (Feedrate Increase) allows moving a series of points that make up a spline in these G-Code Commands: G25, G26, G27.
   a. Press the Spline key to start the spline command.
   b. Enter as many points as needed around the curve.
   c. Press the Spline key to end the spline command and return to Linear mode.

3. **Arc** (Feedrate Decrease) allows an arc to be specified in these G-Code Commands: G02 for clockwise, G03 for counter-clockwise. The Arc will be calculated and presented. After establishing the 3 points of the Arc, the system will return to Linear mode.
   a. Enter 3 points: First, Middle, and End.
   b. Press the Arc key to start the arc command.
   c. Jog to the first point and press Enter .
   d. Jog to the midpoint and press Enter .
   e. Jog to the end point and press Enter .

4. **Back Up** (Spindle RPM Decrease) moves the machine back up to the last Pen Down (PD) point entered. Typically 4 points are saved in the point buffer, which only saves PD moves, to allow for backing up and for generating corners.

5. **Set Home** (Set Home) sets the Soft Home to the current position. The current point will become 0,0.

6. **Go Home** (Go Home) moves to the current Soft Home position.

7. **Close Contour** (Drive Enable {3}) moves to the start of the last PD. A contour is defined as a complete PD to Pen Up (PU) sequence.

8. **2D/3D** (Move {7}) toggles between 2D and 3D: 2D sets X and Y while 3D sets X, Y, and Z. Currently, only 2D is available.

9. **PD/PU** (Set Max Depth) toggles between PD and PU mode in these G-Code Commands: M11 for 2D PD, M21 for 2D PU, M12 for 3D PD, M22 for 3D PU. A point will not be issued at this time as this command only toggles the state. The PD or PU command will be generated on the following move. This allows the operator to produce only one command even if the PD/PU key is pressed multiple times.

10. **Exit** (Cancel/Exit) ends the Job and exits Digitizing in the G-Code Command M02. If the current PD/PU state is PD, then an M21 will be issued to lift the pen prior to exiting.

11. **Enter** (Enter) enters a point. When entering a point in Linear mode, either a G00 or a G01 will be output. The status of the PD or PU will determine which command is generated. A G27 will be output in Spline mode.

12. **Flush Buffer** (Spindle RPM Increase) outputs the current point buffer to the Host PC. When setting points, the operator can save several points in a point buffer prior to being sent so they are not immediately visible in the Digitize Window. The Flush Buffer key will flush this buffer, outputting all points in the buffer so they can be viewed in the Digitize Window. It is not necessary to flush the buffer as the system will automatically do this at the end of the job. Menu

Manual Digitize

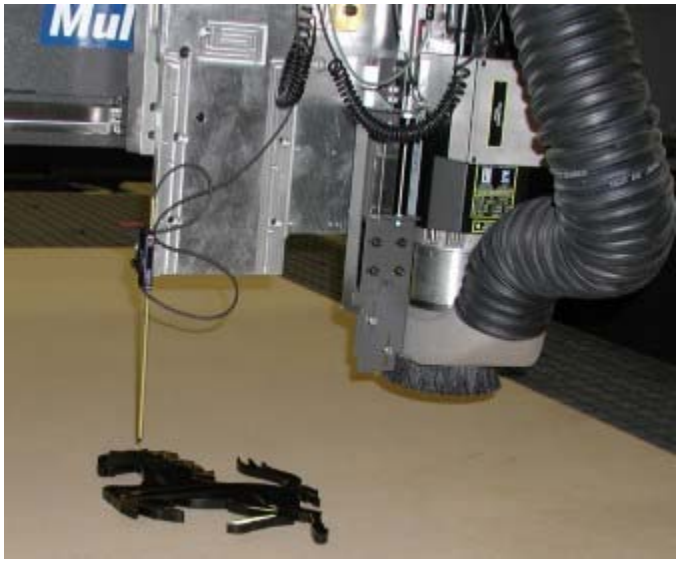
The Manual Digitize function creates a G-Code file one line at a time. When manually digitizing, operators must be sure to enter every point required without overlooking the first Pen Down (PD) move or the final Pen Up (PU) move on a new contour. The job will be either shown in the Digitize window or opened in JobPreviewer. Digitize files can also be imported into a CAD/CAM program or directly run back on the machine.

1. Make sure that JobServer or JobNameServer has been activated on the computer.
2. Press Go Home or move to a location where 0,0 will be located and press Set Home .
3. Press Menu to access the menu system.
4. Press Jog Arrows until the blinking cursor is over the “D” in Digitize.
5. Press Enter to start digitizing.
6. Enter a numerical name for the file to be created. When executing or importing the file, the operator must include a number before “dig.cnc” (e.g., 1234dig.cnc).
7. Jog to the start of a contour and press Enter . This carries the PU move to just above the first location. Digitize starts in a PU mode, so the first move is a PU move.
8. Press Set Max Depth to toggle between PU and PD. This enters a PD move at the same X,Y.
9. Stay in PD mode.
10. Jog around the contour and press Enter to add new points.
11. Press 3 near the end of the contour. This will position the bit directly over the first point in the contour.
12. Press Enter to enter the point.
13. Press Set Max Depth to toggle to PU.
14. Jog to the next contour and repeat the process until all contours have been digitized.
15. Press Go Home to go to 0,0 at the end of the job.
16. Press Exit to end the job.

Menu

Rapid Shape™

The ***MultiCam*** Rapid Shape can automatically digitize both simple and complex shapes. Mounted on any ***MultiCam*** Router, this device enables the router to automatically follow an edge of a pattern or object placed on the router bed and record the outline on the host PC.



Easy and accurate calibration makes digitizing a variety of materials possible with digitizing speeds typically of 60 inches/minute. The Rapid Shape software generates a G-Code file for importing to many CAD/CAM systems.

Menu

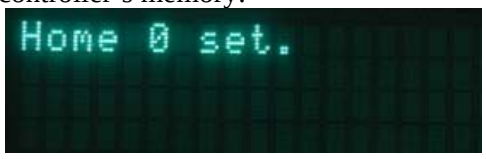
Innovation. Quality. Performance. Best in Industry. Worldwide.

Using Rapid Shape™

Once JobServer or JobNameServer has been verified on the system, Rapid Shape can be used in conjunction with the cutting sequence.

1. Make sure JobServer or JobNameServer is running on the computer.
2. Press Go Home or move to a location where 0,0 will be located and press Set Home .
3. Press Menu to access the menu system.
4. Press Jog Arrows until the blinking cursor is over the “D” in Digitize.
5. Press Enter to access the Digitize sub-menu.
6. Set the 5 options for Rapid Shape: Scan Rate, Step Size, Scan Mode, Add Corners, and Output Start.
   a. **Scan rate** sets the speed the system will auto scan. The default setting is 2.0” per second.
   b. **Step Size** sets the distance along a line between 2 scanned points. The default setting is 0.15”.
   c. **Scan Mode** sets the style of scanning to be completed. Operators should select EDGE for templates and LINE for drawings or patterns.
   d. **Add Corners** adjusts for corners when detected by the system.
   e. **Output START** places “START” at the beginning of the G-Code file, which is required by some software packages due to the importance of the G-Code file.
7. Move the cursor to “Start Digitize” and press Enter .
8. Enter a numeric file name and press Enter .
9. Press Jog Arrows to place the **center of the bit** over the front right corner of the material.
10. Press Set Home .

11. Press Enter to set the home position. This will store the absolute X and Y coordinate locations in the controller’s memory.



12. Press Jog Arrows to move the spindle to the edge of the pattern or template. This sets the direction Rapid Shape will go around the contour of the pattern or template.

13. Press Five to start the Auto Scan. The system will move back and forth for a few seconds to calibrate and then start moving in the set direction along the contour of the pattern. The scanner will continue along the contour until the start point is reached, unless the operator interrupts the sequence.

    a. Press Pause to stop the scanning process if needed.

    b. Press Enter to access manual mode. The operator may continue Auto Scan by moving the spindle forward along the contour at least the step-size distance, while keeping the laser dot close to but not actually on the material, and pressing Auto Scan. The system will move back and forth to calibrate again before continuing around the contour. At the end of the Auto Scan, Digitize will be left in a PU move.

14. Repeat for any additional contours. The system will enter as many contours as needed.

15. Press Cancel to exit out of Digitize mode once the part is completely scanned. To display the part on the computer screen, the operator can open the file in JobPreviewer. The scanned file will be displayed in the preview window.

    Once the scanned file has been completed, it can then be either imported into a CAD/CAM package or executed at the machine through DNC, though the system will **not** complete tool offsets if the operator executes the scanned file from the machine.

Menu

Rapid Shape™ Tips and Troubleshooting

1. **Scan rate** sets the speed the system will auto scan. The default setting is 2.0" per second. A slower scan rate will result in slightly better accuracy at a longer time, while a faster rate will take less time but have less accuracy.

2. **Step Size** sets the distance along a line between 2 scanned points. The default setting is 0.15". The step-size distance should be small for detailed parts but can be larger for parts with less detail.

3. **Scan Mode** sets the style of scanning to be completed. Operators should select EDGE for templates and LINE for drawings or patterns.

4. **Add Corners** can be adjusted for corners when corners are detected during the cutting sequence.

5. **Output START** places "START" at the beginning of the G-Code file, which is required by some software packages due to the importance of the G-Code file.

Rapid Shape will exit if one of the following occurs:

* **The module is not loaded.**
  The operator should load the module by selecting JobServer or JobNameServer from the ***MultiCam*** folder under the Windows Start menu. Once the module is loaded, the router should be calibrated to the material to be scanned.

* **Rapid Shape found the start of the contour.**
  The operator should move to the next contour to continue or cancel the sequence.

* **The operator pressed** Pause .
  The operator can move farther along the contour and restart.

* **The system could not find the edge.**
  The operator can move farther along the contour and restart.

* **The system lost a good edge.**
  The operator can move farther along the contour and restart.

* **Rapid Shape has too many points.**
  The operator can decrease the step size and start over.

* **The previous point is too close to Auto Scan.**
  The operator can move farther along the contour and restart.

Menu

Shutdown

The operator may select the Shutdown function from the menu items to safely shut the machine down.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “S” in Shutdown.

3. Press Enter
to access the Shutdown sequence.

4. Press Enter
to shut down the machine. The system will disable the motor drives and prompt the operator when it is safe to power down the router.

Menu

Cut_Utils

The Cut Utilities function provides the operator with access to utilities to cut material into common shapes, surface or resurface material that needs to be flat, test the output of the system, and cut off the remainder of the material that is presently being cut.

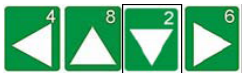
1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “C” in Cut_Utils.

3. Press Enter
to access the Cut Utilities sub-menu.

4. Press Jog Arrows 

until the blinking cursor is over the appropriate option.

Tabl_Mill Matl_Cut
Test_Cut RipCut
Circle Polygon
Rectangle Rect_Rnd
Square

Menu

Table Mill

The Table Mill function is used to surface or resurface the tabletop or any other piece of material that needs to be flat. The program is designed to use any fly cutter. While a 1.25” cutter is supplied with the table, any size cutter can be used. The Table Mill function will surface the table in a rectangular area defined between the Set Home position and the position of the spindle when the Table Mill function is selected. Using the present Set Home, Cut Depth, X & Y Feedrate, Z Plunge Feed, and Tool Lift, the Table Mill function will also turn the spindle and the mist lubrication on and off if applicable.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed).

2. Set home in the lower right hand corner of the table or the material to be milled.

3. Press Jog Arrows to position the cutter in the upper left hand corner of the table or the material to be milled.

4. Press Menu to access the menu system.

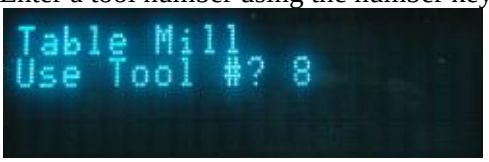
5. Press Jog Arrows until the blinking cursor is over the “C” in Cut_Utils.

6. Press Enter to enter the Cut Utilities sub-menu.

7. Press Jog Arrows until the blinking cursor is over the “T” in Tabl_Mill.

8. Press Enter access the Table Mill function.

9. Enter a tool number using the number keypad and press Enter .



10. Enter the tool diameter using the number keypad if a different size is required. The default value is 1.25” (31.75mm), but the value can be changed for a different size cutter.

11. Press Enter to accept the value. Operators may press X-axis Down Jog Arrow to change Tool Parameters if necessary.

12. Press Start to execute Table Mill, Cancel to abort the function, or Pause to reposition the spindle start point.

The spindle will start, and the machine will begin to mill the table surface. The overlap between passes is 20% of the tool diameter.

Cut_Utils Menu

Table Mill Tool Parameters

The Table Mill Tool Parameters function is used to specify the tool diameter and add lead ins and lead outs.

1. Press X-axis Down Jog Arrow from the Table Mill Start/Cancel/Pause screen.

2. Press Enter to adjust the Table Mill Tool Parameters.

a. Tool Diameter

i. Press Enter over the “T” in Tool Diam.
ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
iii. Press Enter to confirm the value and return to the Table Mill Tool Parameters sub-menu.

b. Lead In/Out

i. Press Enter over the “L” in Lead In.
ii. Press Enter to toggle between YES and NO.

c. Lead In Radius

i. Press Enter over the “L” in Lead In Rad.
ii. Enter the Lead In Radius value using the number keypad.
iii. Press Enter to confirm the value and return to the Table Mill Tool Parameters sub-menu.

Cut_Utils Menu

Test Cut

The Test Cut function is used to run a sample file to test the output of the system. A 0.25” end mill or small engraving cutter can be used to execute the test cut file. If a 0.25” bit is used for the test cut, then the machine will cut a 0.5” circle, 1” diamond, and a 1.5” square. Using the present Set Home, Cut Depth, X & Y Feedrate, Z Plunge Feed, and Tool Lift, the Test Cut function will also turn the spindle and the mist lubrication on and off if applicable.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and make sure the spindle is enabled.

2. Press Menu

to access the menu system.

3. Press Jog Arrows

until the blinking cursor is over the “C” in Cut_Utils.

4. Press Enter to access the Cut Utilities sub-menu.

5. Press Jog Arrows

until the blinking cursor is over the “T” in Test_Cut.

6. Press Enter to access the Test Cut function. Operators may press X-axis Down Jog Arrow

to change Tool Parameters if necessary.

7. Press Start to execute Test Cut, Cancel to abort the function, or Pause to reposition the spindle start point.

Cut_Utils Menu

Test Cut Tool Parameters

The Test Cut Tool Parameters function is used to specify the tool diameter and to add lead ins and lead outs.

1. Press X-axis Down Jog Arrow from the Test Cut Start/Cancel/Pause screen.

2. Press Enter to adjust the Test Cut Tool Parameters.

a. Tool Diameter

i. Press Enter over the “T” in Tool Diam.
ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
iii. Press Enter to confirm the value and return to the Test Cut Tool Parameters sub-menu.

b. Lead In/Out

i. Press Enter over the “L” in Lead In.
ii. Press Enter to toggle between YES and NO.

c. Lead In Radius

i. Press Enter over the “L” in Lead In Rad.
ii. Enter the Lead In Radius value using the number keypad.
iii. Press Enter to confirm the value and return to the Test Cut Tool Parameters sub-menu.

Cut_Utils Menu

Circle

The Circle function is used to run a Circle Shape sample file to test the output of the system. A 0.125” end mill or small engraving cutter can be used to execute the test cut file. Using the present Set Home, Cut Depth, X & Y Feedrate, Z Plunge Feed, and Tool Lift, the Circle function will also turn the spindle and the mist lubrication on and off if applicable.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and make sure the spindle is enabled.

2. Press Menu

to access the menu system.

3. Press Jog Arrows

until the blinking cursor is over the “C” in Cut_Utils.

4. Press Enter to access the Cut Utilities sub-menu.

5. Press Jog Arrows

until the blinking cursor is over the “C” in Circle.

6. Press Enter to access the Circle function.

7. Enter the radius of the Circle on the display using the number keypad (e.g., 0.50) and press Enter .

Operators may press X-axis Down Jog Arrow

to change Tool Parameters if necessary.

8. Press Start to execute Circle Cut, Cancel to abort the function, or Pause to reposition the spindle start point.

Cut_Utils Menu

Circle Shape Tool Parameters

The Circle Shape Tool Parameters function is used to specify the tool diameter and to add lead ins and lead outs.

1. Press X-axis Down Jog Arrow from the Circle Shape Start/Cancel/Pause screen.

2. Press Enter to access the Circle Shape Tool Parameters.

a. Tool Diameter

i. Press Enter over the “T” in Tool Diam.
ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
iii. Press Enter to confirm the value and return to the Circle Shape Tool Parameters sub-menu.

b. Lead In/Out

i. Press Enter over the “L” in Lead In.
ii. Press Enter to toggle between YES and NO.

c. Lead In Radius

i. Press Enter over the “L” in Lead In Rad.
ii. Enter the Lead In Radius value using the number keypad.
iii. Press Enter to confirm the value and return to the Circle Shape Tool Parameters sub-menu.

Cut_Utils Menu

Rectangle

The Rectangle function is used to run a Rectangle Shape sample file to test the output of the system. A 0.25” end mill or small engraving cutter can be used to execute the test cut file. Using the present Set Home, Cut Depth, X & Y Feedrate, Z Plunge Feed, and Tool Lift, the Rectangle function will also turn the spindle and the mist lubrication on and off if applicable.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and make sure the spindle is enabled.
2. Press Menu

to access the menu system.
3. Press Jog Arrows

until the blinking cursor is over the “C” in Cut_Utils.
4. Press Enter to access the Cut Utilities sub-menu.
5. Press Jog Arrows

until the blinking cursor is over the “R” in Rectangle.
6. Press Enter to access the Rectangle function.
7. Enter the length of the X and Y sides for the Rectangle using the number keypad (e.g., X=19, Y=20).

8. Press Enter to accept the value.

9. Press Enter to accept the value. Operators may press X-axis Down Jog Arrow

to change Tool Parameters if necessary.

10. Press Start to execute Rectangle Cut, Cancel to abort the function, or Pause to reposition the spindle start point.

Cut_Utils Menu

Rectangle Shape Tool Parameters

The Rectangle Shape Tool Parameters function is used to specify the tool diameter and to add lead ins and lead outs.

1. Press X-axis Down Jog Arrow from the Rectangle Shape Start/Cancel/Pause screen.

2. Press Enter to adjust the Rectangle Shape Tool Parameters.

a. Tool Diameter

i. Press Enter over the “T” in Tool Diam.
ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
iii. Press Enter to confirm the value and return to the Rectangle Shape Tool Parameters sub-menu.

b. Lead In/Out

i. Press Enter over the “L” in Lead In.
ii. Press Enter to toggle between YES and NO.

c. Lead In Radius

i. Press Enter over the “L” in Lead In Rad.
ii. Enter the Lead In Radius value using the number keypad.
iii. Press Enter to confirm the value and return to the Rectangle Shape Tool Parameters sub-menu.

Cut_Utils Menu

Square

The Square function is used to run a Square Shape sample file to test the output of the system using the present Set Home, Cut Depth, Tool Lift, X & Y Feedrate, and Z Plunge Feed.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed).
2. Press Menu

to access the menu system.
3. Press Jog Arrows

until the blinking cursor is over the “C” in Cut_Utils.
4. Press Enter to access the Cut Utilities sub-menu.
5. Press Jog Arrows

until the blinking cursor is over the “S” in Square.
6. Press Enter to access the Square function.
7. Enter the length of the side for the Square using the number keypad (e.g., 9.000).

8. Press Enter to accept the value. Operators may press X-axis Down Jog Arrow

to change Tool Parameters if necessary.

9. Press Start to execute Square Cut, Cancel to abort the function, or Pause to reposition the start point.

Cut_Utils Menu

Square Shape Tool Parameters

The Square Shape Tool Parameters function is used to specify the tool diameter and add lead ins and lead outs.

1. Press X-axis Down Jog Arrow from the Square Shape Start/Cancel/Pause screen.

2. Press Enter to adjust the Square Shape Tool Parameters.

a. Tool Diameter

i. Press Enter over the “T” in Tool Diam.
ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
iii. Press Enter to confirm the value and return to the Square Shape Tool Parameters sub-menu.

b. Lead In/Out

i. Press Enter over the “L” in Lead In.
ii. Press Enter to toggle between YES and NO.

c. Lead In Radius

i. Press Enter over the “L” in Lead In Rad.
ii. Enter the Lead In Radius value using the number keypad.
iii. Press Enter to confirm the value and return to the Square Shape Tool Parameters sub-menu.

Cut_Utils Menu

Material Cutoff

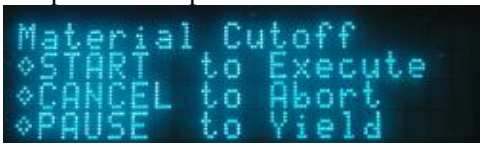
The Material Cutoff function will allow the operator to manually cut off the remainder of the material that is presently cutting in a single, straight-line or diagonal cutting move. Using the present Cut Depth, X & Y Feedrate, Z Plunge Feed, and Tool Lift, the Material Cutoff function will also turn the spindle and the mist lubrication on and off if applicable.

1. Set up the system as if a normal file is to be executed (e.g., Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed). It is not necessary to set home as the cut will begin at the current X,Y location of the spindle.
2. Press Jog Arrows to position the cutter in the desired start position.
3. Press Menu to access the menu system.
4. Press Jog Arrows until the blinking cursor is over the “C” in Cut_Utils.
5. Press Enter to access the Cut Utilities sub-menu.
6. Press Jog Arrows until the blinking cursor is over the “M” in Matl_Cut.
7. Press Enter to access the Material Cutoff function.
8. Enter the desired tool number if different from the one currently loaded and press Enter .

9. Enter the desired value for the X Cutoff Distance using the number keypad (e.g., 0) and press Enter .

10. Enter the desired value for the Y Cutoff Distance using the number keypad (e.g., 60) and press Enter .

11. Press Start to execute Material Cutoff, Cancel to abort the function, or Pause to reposition the spindle start point.

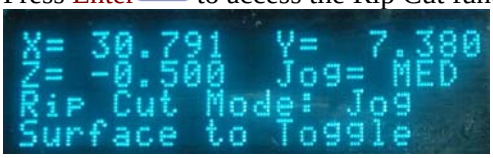


Cut_Utils Menu

Rip Cut

The Rip Cut function will allow the operator to manually cut wherever the machine is Jogged so that the operator can remove excess material or even cut out simple shapes if desired. Using the present Cut Depth, X & Y Feedrate, Z Plunge Feed, and Tool Lift, the Rip Cut function will also turn the spindle and the mist lubrication on and off if applicable.

1. Set up the system as if a normal file is to be executed (e.g., Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed). It is not necessary to set home as the cut will begin at the current X,Y location of the spindle.
2. Press Jog Arrows to position the cutter in the desired start position.
3. Press Menu to access the menu system.
4. Press Jog Arrows until the blinking cursor is over the “C” in Cut_Utils.
5. Press Enter to access the Cut Utilities sub-menu.
6. Press Jog Arrows until the blinking cursor is over the “R” in RipCut.
7. Press Enter to access the Rip Cut function.



8. Press Set Surface to toggle between Jog and Cut.

9. Press Jog Arrows to start moving the spindle, which will plunge into the material after reaching the programmed RPM. The spindle will cut for as long as the Jog button is depressed.
10. Press Set Surface again and as needed to toggle between Jog and Cut
11. Press Cancel to exit the Rip Cut mode.

Cut_Utils Menu

Polygon

The Polygon function is used to run a Polygon Shape sample file to test the output of the system. A 0.125” end mill or small engraving cutter can be used to execute the test cut file. Using the present Set Home, Cut Depth, X & Y Feedrate, Z Plunge Feed, and Tool Lift, the Polygon function will also turn the spindle and the mist lubrication on and off if applicable.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and verify that the spindle is enabled.
2. Press Menu

to access the menu system.
3. Press Jog Arrows

until the blinking cursor is over the “C” in Cut_Utils.
4. Press Enter to access the Cut Utilities sub-menu.
5. Press Jog Arrows

until the blinking cursor is over the “P” in Polygon.
6. Press Enter to access the Polygon function.

7. Enter the number of sides for the Polygon using the number keypad (e.g., 3) and press Enter .

8. Enter the radius of the Polygon using the number keypad (e.g., 0.5) and press Enter .

Operators may press X-axis Down Jog Arrow

to change Tool Parameters if necessary.

9. Press Start to execute Polygon Cut, Cancel to abort the function, or Pause to reposition the spindle start point.

Cut_Utils Menu

Polygon Shape Tool Parameters

The Polygon Shape Tool Parameters function is used to specify the tool diameter and to add lead ins and lead outs.

1. Press X-axis Down Jog Arrow from the Polygon Shape Start/Cancel/Pause screen.

2. Press Enter to adjust the Polygon Shape Tool Parameters.

a. Tool Diameter

i. Press Enter over the “T” in Tool Diam.
ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
iii. Press Enter to confirm the value and return to the Polygon Shape Tool Parameters sub-menu.

b. Lead In/Out

i. Press Enter over the “L” in Lead In.
ii. Press Enter to toggle between YES and NO.

c. Lead In Radius

i. Press Enter over the “L” in Lead In Rad.
ii. Enter the Lead In Radius value using the number keypad.
iii. Press Enter to confirm the value and return to the Polygon Shape Tool Parameters sub-menu.

Cut_Utils Menu

Radiused Rectangle

The Radiused Rectangle function is used to run a Radiused Rectangle Shape sample file to test the output of the system. A 0.25” end mill or small engraving cutter can be used to execute the test cut file. Using the present Set Home, Cut Depth, X & Y Feedrate, Z Plunge Feed, and Tool Lift, the Radiused Rectangle function will also turn the spindle and the mist lubrication on and off if applicable.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and make sure the spindle is enabled.
2. Press Menu

to access the menu system.
3. Press Jog Arrows

until the blinking cursor is over the “C” in Cut_Utils.
4. Press Enter to access the Cut Utilities sub-menu.
5. Press Jog Arrows

until the blinking cursor is over the “R” in Rect_Rnd.
6. Press Enter to access the Radiused Rectangle function.
7. Enter the length of the X and Y sides for the Rectangle using the number keypad (e.g., X=19, Y=20).

8. Press Enter to accept the value.

9. Press Enter to accept the value.
10. Enter the radius of the corners using the number keypad (e.g., 1.5).

11. Press Enter to accept the value of the radius. Operators may press X-axis Down Jog Arrow to change Tool Parameters if necessary.

12. Press Start to execute Radiused Rectangle Cut, Cancel to abort the function, or Pause to reposition the start point. Cut_Utils Menu

Radiused Rectangle Shape Tool Parameters

The Radiused Rectangle Shape Tool Parameters function is used to specify the tool diameter and add lead ins and lead outs.

1. Press X-axis Down Jog Arrow from the Radiused Rectangle Shape Start/Cancel/Pause screen.

2. Press Enter to adjust the Radiused Rectangle Shape Tool Parameters.

a. Tool Diameter

i. Press Enter over the “T” in Tool Diam.
ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
iii. Press Enter to confirm the value and return to the Radiused Rectangle Shape Tool Parameters sub-menu.

b. Lead In/Out

i. Press Enter over the “L” in Lead In.
ii. Press Enter to toggle between YES and NO.

c. Lead In Radius

i. Press Enter over the “L” in Lead In Rad.
ii. Enter the Lead In Radius value using the number keypad.
iii. Press Enter to confirm the value and return to the Radiused Rectangle Shape Tool Parameters sub-menu.

Cut_Utils Menu

Utility

The Utility sub-menu is used to provide the operator with additional options for completing, changing, or monitoring a cutting sequence. Some of these menu items contain a sub-menu for further specifications.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the appropriate option. Certain menu items will then enter a sub-menu with more selections.

Clr_Home	GDrillTest	
No_Spin	Inv_Reset	
Job_Time	Reboot	
ParkPos	AdjSurf	
SPA Trigg	DNC_Mode	
Lube	Dust Ctrl	
PauseCfg	MisterOpt	Menu

Clr_Home

The Clr_Home function is used to clear the Soft Home location 0 and will reset to the Hard Home position. The Hard Home position will bethe origin (0,0) until a new home is set from the menu system.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “C” in Clr_Home.

5. Press Enter to clear the Home position and return to the main display.

Utility Menu

No_Spin

The No_Spin function is used to disable the spindle motor when the operator is using a vinyl knife, pen, or comparable tool.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “N” in No_Spin.

5. Press Enter to access the No_Spin function.

6. Enter the tool number to be set to no spin. Selecting 0 will disable the function.

7. Press Enter to accept the tool number and return to the Utility menu.

Utility Menu

Job_Time

The Job Time function is used to display the amount of time the previous job required.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.

3. Press Enter
to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “J” in Job_Time.

5. Press Enter
to view the exact job time.

Utility Menu

ParkPos

The ParkPos function is used to set a specific Park location for the X- and Y-axes to assume after each job, which is useful when currently using only a portion of the available table space. The X- and Y-axes can be set to move just far enough out of the way for the material to be removed instead of parking all the way to the rear every time.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “P” in ParkPos.

5. Press Enter to access the Park Position function.

6. Enter the desired park position for the spindle in X or enter 0 to use the table size.

7. Press Enter to accept the position and return to the Utility menu.

Utility Menu

SPA Trigg

The Spindle Amperage Trigger (SPA Trigg) function allows the operator to check the quality of the cutting tool after a set time of consistent cutting so that any changes can be made (e.g., sharpening, alternating) to prevent any unnecessary wear on the machine. The operator has the option to disable this function by entering 0 at the appropriate prompt.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.

3. Press Enter
to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “S” in SPA Trigg.

5. Press Enter
to access the SPA Trigg sub-menu.

6. Position the cursor over the appropriate parameter to set up the SPA Trigg and press Enter .
   a. Enter the maximum current level in amps using the number keypad when setting the SPA Level or enter the time at the max current in seconds when setting the SPA Time.
   b. Disable this function by selecting *SPA Time* and entering 0.

Utility Menu

Lube

The Lube function has 4 separate sub-functions that relate to the lubrication process on machines that have the auto mister installed.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

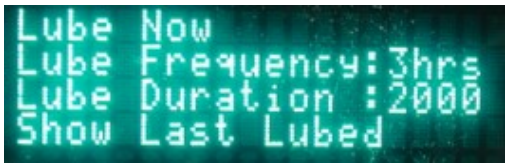
until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “L” in Lube.

5. Press Enter to access the Lube sub-menu.



6. Press Jog Arrows

until the blinking cursor is over the first letter of the appropriate item.

- Lube Now will apply lubrication to the system.
- Lube Frequency will allow the operator to set how often the lubrication should be applied to the system.
- Lube Duration will specify how long the lubrication should be applied (in msec).
- Show Last Lubed will display the last time lubrication was applied to the system.

Utility Menu

PauseCfg

The Pause Configuration function allows operators to disregard the pause called out in the M Code (i.e., M01) usually included for testing purposes.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

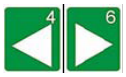
until the blinking cursor is over the “U” in Utility.

3. Press Enter
to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “P” in PauseCfg.

5. Press Enter
to access the Pause Configuration setting.

6. Press Y-axis Jog Arrows 

to toggle between Yes and No.

7. Press Cancel
to exit Pause Configuration.

Utility Menu

GDrillTest

The Gang Drill Test function is used to perform a test of the Gang Drill. If there is no gang drill option on the router, then the Utility sub-menu will not offer this selection.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the GDrillTest menu item.

5. Press Enter to test the Gang Drill.

6. Press Soft Home to toggle the Gang Drill on and off.

7. Press Z-axis Jog Arrows

to raise or lower the drill bank.

8. Press the number keys to fire individual drills by number.

Utility Menu

Inv_Reset

The Inv_Reset function is used to reset the spindle inverter.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “I” in Inv_Reset.

5. Press Enter to reset the spindle inverter and return to the Utility menu.

Utility Menu

Reboot

The Reboot function is used to restart the machine without having to power down the machine.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “R” in Reboot.

5. Press Enter to access the reboot process.

6. Press Enter to process the reboot of the machine.

Utility Menu

AdjSurf

The AdjSurf function is used to adjust the surface setting without having to go through the surfacing routine.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

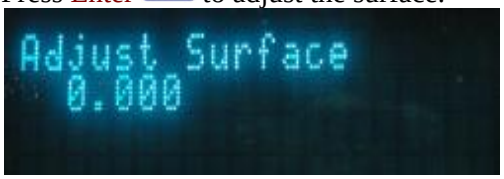
until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

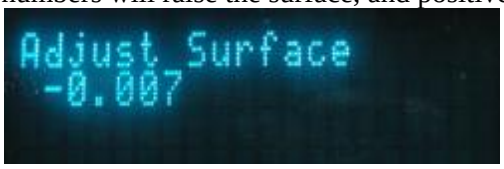
until the blinking cursor is over the “A” in AdjSurf.

5. Press Enter to adjust the surface.

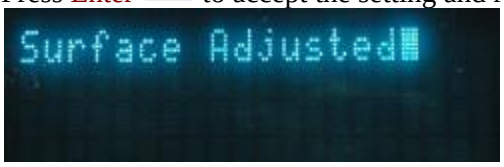


6. Press Z-axis Jog Arrows

to adjust the surface in one-thousandth increments (e.g., 0.001). Negative numbers will raise the surface, and positive numbers will lower the surface.



7. Press Enter to accept the setting and return to the main display.



Utility Menu

DNC Mode

The DNC Mode function is used to allow the operator to toggle back and forth between local and remote DNC Modes. Local mode allows the operator to review what files have been saved in the controller, and remote mode allows the operator to run programs. The system must be set to remote mode to run the file.

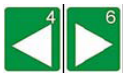
The DNC Mode can create a shortcut for commonly used files without the operator having to enter the Utility sub-menu each time. The controller memory allows a maximum of 20 job files or a limited amount of memory space for large job files to be saved to the controller. The display will indicate when the memory capacity is full with a notice showing *Error Writing File; Out of Room*. The operator will need to delete a job file from the controller memory before adding any new job files.

1. Press Menu

to access the menu system.
2. Press Jog Arrows

until the blinking cursor is over the “U” in Utility.
3. Press Enter to access the Utility sub-menu.
4. Press Jog Arrows

until the blinking cursor is over the “D” in DNC Mode.
5. Press Enter to access the DNC mode. An asterisk will be located to the left of the mode currently being run.

6. Press Y-axis Jog Arrows 

to toggle between the DNC Modes. Local mode shows what files have been saved to the controller while the remote mode shows what files are available in the PC.
7. Press Enter to accept the selected mode and return to the main display.

Saving Files to Controller DNC

1. Make sure the DNC mode is set for local.

2. Press Shift and DNC to access the list of files available in the PC.

3. Press Jog Arrows until the blinking cursor is over the particular job file to be saved to the controller.
4. Press Enter to accept the job file.
5. Press Start to save the file to the controller. The operator can access the file from the DNC function at the keypad.
6. Press Cancel to return to the main display.
7. Press DNC to access the files available from the controller.
8. Press Jog Arrows until the blinking cursor is over the particular job file to be run.
9. Press Enter to access the job file and then Start to execute the job file.

Deleting Files from Controller DNC

1. Make sure the DNC mode is set for local.

2. Press DNC to access the list of job files available in the controller.
3. Press Jog Arrows until the blinking cursor is over the particular job file to be deleted from the controller.
4. Press Enter to access the job file.
5. Press Set Max Depth to set the system to delete the job file.
6. Press Enter to delete the job file.

Utility Menu

Dust Control

The Dust Control function allows the operator to select whether or not the dust collector will operate in conjunction with a specific cutting tool. If the dust collector is disabled on a specific tool, then the dust collector will automatically raise up during the tool loading and remain up during the cutting sequence.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

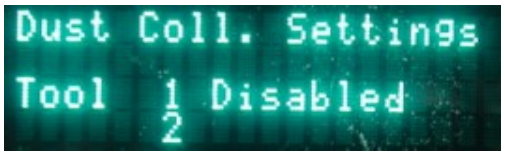
until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “D” in Dust Ctrl.

5. Press Enter to access the dust collector settings.



6. Enter the appropriate tool number and press Enter to disable the dust collector. The dust collector will remain in an up position while the tool cuts the material. Operators may undo the disabled status once the cutting sequence has completed.

Utility Menu

MisterOpt

The Mister Option function allows operators to disable the auto mister during the job file. Operators will need to enable the Mister Option function for the machine to run the auto mister during any following job files.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

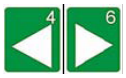
until the blinking cursor is over the “U” in Utility.

3. Press Enter to access the Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “M” in MisterOpt.

5. Press Enter to disable the auto mister.

6. Press Y-axis Jog Arrows 

to toggle between Yes and No.

7. Press Enter to save the selection and exit Mister Option.

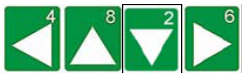
Utility Menu

Tool_Comp

The Tool_Comp function is used to select a particular compensation value for a specific tool.

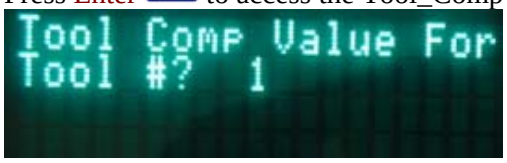
1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “T” in Tool_Comp.

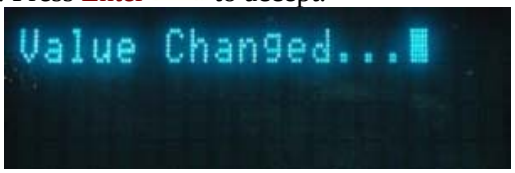
3. Press Enter to access the Tool_Comp function.



4. Enter the correct tool using the number keypad (e.g., 1) and press Enter .

5. Enter a tool compensation value that is equal to or less than 0.10 (e.g., 0.10).

6. Press Enter to accept.



Menu

Vision

MultiVision offers operators the option in completing various projects to use cameras for setting up fiducials, or reference marks for Vision Tool software, to streamline repetitive cutting projects of similar material or design. The Vision menu offers several different options for operators to utilize their system for an improved output. MultiVision offers a few settings that must be set once and then are stored in the controller memory until those settings are overwritten.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the appropriate option. Certain menu items will then enter a sub-menu with more selections.

Focus Camera
Fiducial Diameter
Correlation
Retake Distance
Settling Time
Flip to Cut
Auto Focus
Fast Find
Camera Delay
Auto Find Setting
Manual Find Setting
Set Camera Scale
Set Camera Offsets
Move to Cam Offsets

Menu

Focus Camera

The Focus Camera function is used to set the system to look for the fiducial, calibrate the tool if needed, and focus the camera. This enables the file to run accurately. The camera must be focused each time the material thickness changes on a standard machine. Operators should always leave the table vacuum on and use a computer that can be seen from the machine during the focusing process.

1. Calibrate the 0.25” (6.35mm) End Mill and surface the tool to the top of the material.

2. Press Menu

to access the menu system.

3. Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

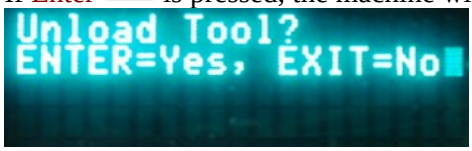
4. Press Enter to access the Vision sub-menu.

5. Press Jog Arrows

until the blinking cursor is over the “F” in Focus Camera.

6. Press Enter to unload the tool or Cancel to leave the tool in the spindle if prompted. The tool may need to be unloaded depending on the length of the tool. Longer tools may interfere with the focusing process.

   If Enter is pressed, the machine will unload the tool and return to the material.

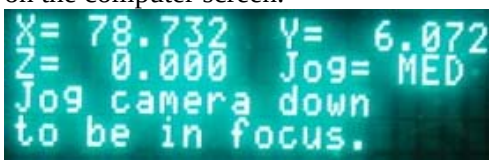


7. Press Jog Arrows

to reach the fiducial on the material to be cut.

8. Jog Knife 1 (i.e., Z2) using the Z-axis Jog Arrows

at the prompt to bring the fiducial into focus on the computer screen.



9. Press Enter when the fiducial is in focus.

Vision Menu

Fiducial Diameter

The Fiducial Diameter function is used to inform the system as to the size of the fiducial when the system is searching. This diameter can be set up in either inches or millimeters.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

3. Press Enter
to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “F” in Fiducial Diameter.

5. Press Enter
to adjust the Fiducial Diameter.

6. Enter the value using the number keypad (e.g., 0.250).

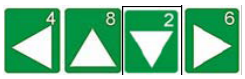
7. Press Enter
to save the Fiducial Diameter and return to the Vision menu. Vision Menu

Correlation

The Correlation function is used to establish the pass/fail values for settings that are typically in the range of 0.2 to 1.0. This function can be utilized if the material to be cut has a fuzzy or indistinct fiducial and the camera is unable to capture a clear image. If the correlation value is too low or too high, it will be rejected.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “C” in Correlation Value.

5. Press Enter to access the Correlation Value function.

6. Enter the correlation value using the number keypad (e.g., 0.200).

7. Press Enter to save the value and return to the Vision menu.

Vision Menu

Retake Distance

The Retake Distance function allows the operator to improve the accuracy of the picture. The system will calculate how far away the fiducial is when a picture is taken. If that distance is greater than the retake distance, the camera will center the fiducial and take another picture.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “R” in Retake Distance.

5. Press Enter to access the Retake Distance function.

6. Enter the distance for the camera to adjust so that the fiducial can be pictured clearly (e.g., 0.2” or 0.127mm).

7. Press Enter to save the distance and return to the Vision menu. Vision Menu

Settling Time

The Settling Time function is used to specify how long the system must wait before reading the fiducial. This is helpful in case the camera is shaking after slewing into position. Most systems work best with 20ms. The Settling Time function applies before the picture is taken while the Camera Delay function applies after the picture is taken.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “S” in Settling Delay.

5. Press Enter to adjust the Settling Delay.

6. Enter the value using the number keypad (e.g., 200msec).

7. Press Enter to save the delay time and return to the Vision menu.

Vision Menu

Flip to Cut

The Flip to Cut function is used to complete cutting sequences for both sides of the material. Operators who require different types of cuts on either side of the material can run the file, flip the material, and finish the cutting sequence. In order to properly align the material for the reverse cut, operators will need to have pop-up pins installed on their machines.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “F” in Flip to Cut.

5. Press Enter to toggle between No and Yes.

Vision Menu

Auto Focus

The Auto Focus function is used to enable or disable the automatic focusing function of the MultiVision camera. Operators can set the system to automatically use the size of the fiducial to determine if the fiducial is in focus. If the size of the fiducial is off by more than 10%, then the camera will raise or lower until the size is within 10%.

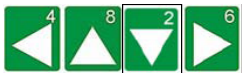
1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows 

until the blinking cursor is over the “A” in Auto Focus.

5. Press Enter to toggle between No and Yes.

Vision Menu

Fast Find

The Fast Find function is used to enable the camera to begin the rapid search for the remaining fiducials. Once enabled, the Fast Find function allows the camera to take a picture of the fiducial, wait for the camera delay, and then start to move to the next fiducial before the data from the camera is returned. This value is typically set to 2 since, after locating the second fiducial, the system is able to identify the rotation and offset to find the remaining fiducials.

- Operators may disable the Fast Find function by **setting the value to 0**, which would be recommended for first runs of a job file or for material that is stretched or skewed.
- Operators in production runs who are able to place the material so that the second fiducial is always in the FOV can **set Fast Find to 1**.
- If Auto Find is enabled on the system, then **Fast Find will be disabled until the third fiducial**.
- If the fiducial is not found while the system is in Fast Find mode, then the camera will move back to the **previous fiducial to take another picture**.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “F” in Fast Find.

5. Press Enter to access the Fast Find function.

6. Input the fiducial number and press Enter . The value will be on the display.

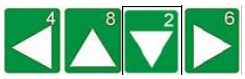
Vision Menu

Camera Delay

The Camera Delay function sets how long the system waits to move after taking a picture of the fiducial. If the value is too low, then the operator will see blurry images in Vision Tool. The Settling Time function applies before the picture is taken while the Camera Delay function applies after the picture is taken.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “C” in Camera Delay.

5. Press Enter to access the camera delay.

6. Input a value for the delay and press Enter . The display will show the amended value.

Vision Menu

Auto Find Setting

The Auto Find Setting function is used to specify the distance that the system will scan for the fiducial before prompting the operator to Jog to the proper location if the fiducial is out of range. Auto Find Settings will prompt the operator to Jog to the area where the first fiducial should be without trying to auto find it and will only search for the second and later fiducials by looking in the area of where they should be.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “A” in Auto Find Setting.

5. Press Enter to access the Auto Find Setting function.

6. Enter the distance for the camera to scan for the fiducial (e.g., 0.2” or 0.127 mm) or leave as “0” so the operator may manually locate the second and third fiducials.

7. Press Enter to save the distance and return to the Vision menu.

Vision Menu

Manual Find Setting

The Manual Find Setting function can be utilized once the Auto Find Setting has been disabled. Operators can enter “0” under Auto Find Setting before selecting Manual Find Setting and then select to either Jog the tool to the fiducial or Jog the camera to the fiducial. Operators can press Enter once the tool bit or the camera is directly above the fiducial.

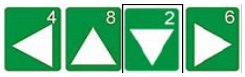
1. Press Menu to access the menu system.
2. Press Jog Arrows until the blinking cursor is over the “V” in Vision.
3. Press Enter to access the Vision sub-menu.
4. Press Jog Arrows until the blinking cursor is over the “M” in Manual Find Setting.
5. Press Enter to access the Manual Find Setting function.

6. Press X-axis Jog Arrows to move between “Jog Tool to Fid.” and “Jog Camera to Fid.” The double arrows show which setting is active.
7. Press Enter to return to the Vision menu.

Vision Menu

Set Camera Scale

The Set Camera Scale function is used to calibrate the camera scale. The operator specifies a distance to which the machine will move, take a picture, move back, take a picture, calculate the scale, and save the camera scale. The camera scale can be set at the initial setup.

1 Press Jog Arrows  to reach a fiducial.

2. Press Menu to access the menu system.

3. Press Jog Arrows until the blinking cursor is over the “V” in Vision.

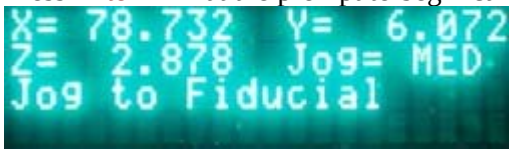
4. Press Enter to access the Vision sub-menu.

5. Press Jog Arrows until the blinking cursor is over the “S” in Set Camera Scale.

6. Press Enter to access the Set Camera Scale function. The keypad display will prompt the operator for a move size. The operator specifies a distance to which the machine will move, take a picture, move back, take a picture, calculate the scale, and save the camera scale.

7. Enter a value of 0.2 for 1” FOV lens (0.127 for 6.35 mm) and press Enter . The camera will complete a series of 0.2” (0.127 mm) moves off the fiducial point as the camera checks for pixels per inch (mm). The scale will be based on these readings. A typical scale value will be 1000 ± 100 for a 1” (25.4mm) field of view.

8. Press Enter at the prompt to begin calibration.



9. Press Jog Arrows to reach the fiducial on the material to be cut and press Enter .

Vision Menu

Set Camera Offsets

The Set Camera Offsets function is used to set the camera offsets based on the center of the spindle and the center of the camera by marking a hole with the Cut Depth entered in the Params_2D menu. The system prompts the operator to Jog over until the fiducial is close to the center of the computer screen. The location does not have to be exact. Camera offsets are usually set at installation and do not need to be reset unless the spindle or camera is replaced.

1. Spray paint a portion of the spoilboard or any scrap material and allow the paint to dry.

2. Set surface on the spray painted section using a 0.25” (6.35mm) End Mill.

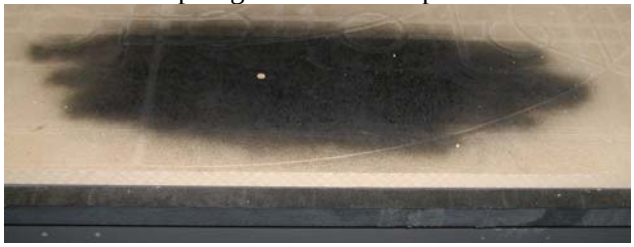
3. Set Home in the spray painted section.

4. Set the Cut Depth from the Params_2D menu to cut into the spoilboard. Cut thicknesses may vary for each customer, though 0.01 – 0.02” (0.25 – 0.51mm) is typical.
   a. Open Cut Depth from the Params_2D menu.

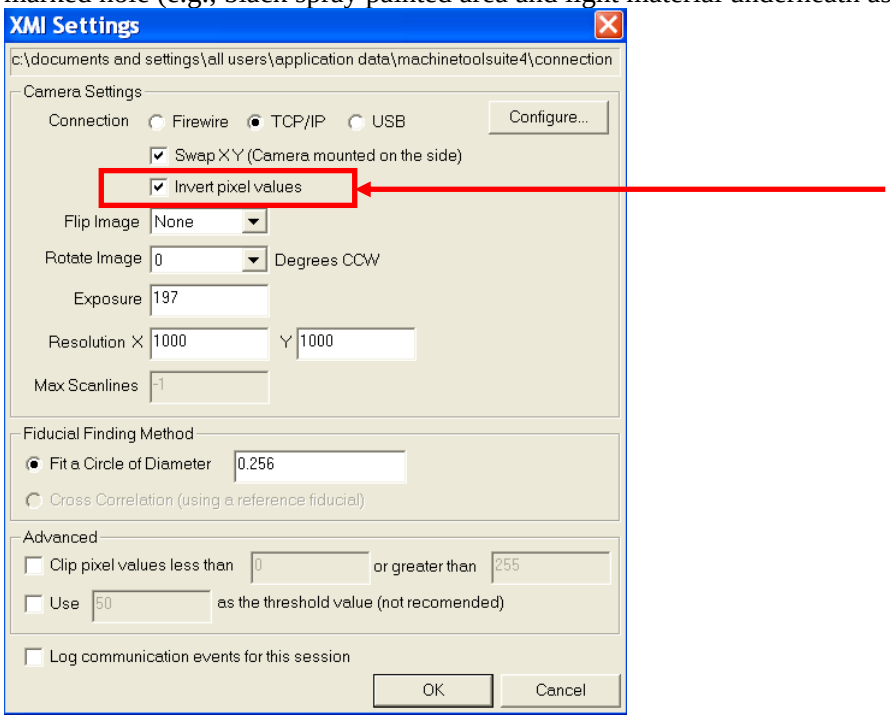
   b. Enter the appropriate thickness using the number keypad (e.g., 0.020” [0.51mm]).
   c. Press Cancel to exit the Params_2D menu.

5. Set the Camera Offsets.
   a. Open Setting Camera Offsets from the Vision menu.

   b. Press Enter to allow the spindle to mark a fiducial at the current home. The spindle will turn on, and the router will plunge to the Cut Depth.



6. Open VisionTool check the box next to *Invert pixel values* when the contrast material is darker than the marked hole (e.g., black spray painted area and light material underneath as shown above).



7. Select *OK*.

8. Jog the camera until the fiducial is in the center of the VisionTool window on the computer screen.

9. Review the keypad display showing the offset amount and offset of fiducial and press Enter . This will save the offsets.

   The controller will save the camera offsets in Flash parameters **132** (X-axis offset) and **133** (Y-axis offset). The camera offsets will be saved in the computer until overwritten or deleted by the operator and do not need to be reset for each job.

10. Uncheck the box next to *Invert pixel values* and select *OK*.

Vision Menu

Move to Cam Offsets

The Move to Camera Offsets function is mainly used as a verification tool to center the camera over the fiducial.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

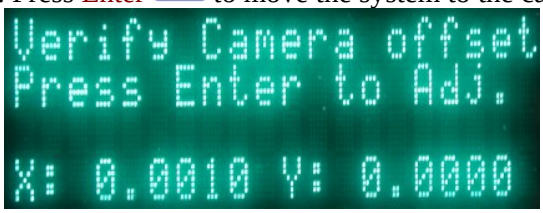
until the blinking cursor is over the “V” in Vision.

3. Press Enter to access the Vision sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “M” in Move to Cam Offsets.

5. Press Enter to move the system to the camera offsets.



6. Click on Live Feed in the toolbar.

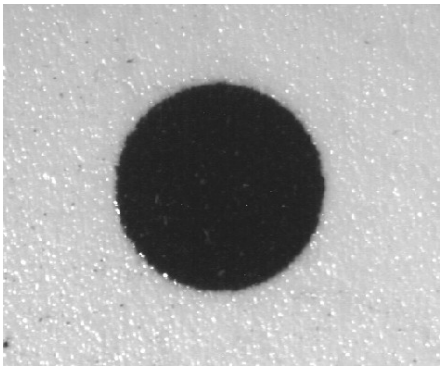
7. Press Jog Arrows

to move the camera over a fiducial.

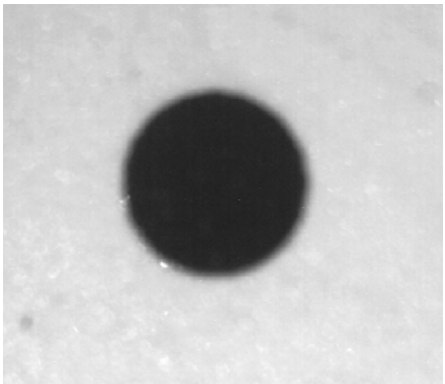
8. Press Z-axis Jog Arrows

to bring the fiducial into clearer view.

Fiducial in Focus



Fiducial out of Focus



9. Press Enter once the fiducial is clear and return to the Vision menu. Vision Menu

ATC

The Automatic Tool Changer (ATC) sub-menu is used to calibrate, adjust, load, and unload tools and is only available on machines with the ATC option.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “A” in ATC.

3. Press Enter
to access the ATC sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the appropriate option.

Cal_Tool
Adj_Tool
Tool_Unld
Get_Tool
CalGDrill
LoadDrill
AdjGDrill
CfgGDrill
LoadTools

Menu

Tool Calibration

The Tool Calibration function is used to calibrate new tools when a new tool is inserted into the ATC. The machine will store the Z-axis tool offsets of the new tool, and this allows for fast tool changes. The Z-axis tool offset of each tool is stored in memory even after the machine is turned off.

1. Make sure the tool to be calibrated has been properly loaded in the ATC.
2. Press Menu

to access the menu system.
3. Press Jog Arrows

until the blinking cursor is over “A” in ATC.
4. Press Enter
to access the ATC sub-menu.
5. Press Jog Arrows

until the blinking cursor is over the “C” in Cal_Tool.
6. Press Enter
to access the Tool Calibration function.
7. Select the number of the tool to calibrate using the number keypad (e.g., 1).
8. Press Enter
. The spindle will move over to the tool carousel, unload the tool if one is currently in place, rotate to the selected tool, and load that tool into the system. The system will then move over the calibration block.
9. Press the Z-axis Jog Arrows

to lower the spindle until it is near the top of the calibration block.
10. Place the grounding cradle on the ATC mounting groove in the spindle tool holder.
11. Press and hold Zero

to begin the tool calibration. Once Zero

is pressed, the Z-Axis will move down slowly until it senses the calibration block. It will then lift off the block up to maximum Z height.
12. Remove the mounting collar before beginning work.

ATC Menu

Tool Unload

The Tool Unload function allows the ATC to function as a non-Tool Changer. This is helpful when running a job that uses 1 tool for a long period of time.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

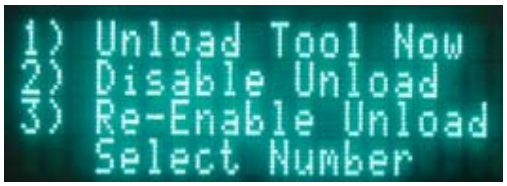
until the blinking cursor is over the “A” in ATC.

3. Press Enter
to access the ATC sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “T” in Tool_Unld.

5. Press Enter
to unload a tool.



6. Enter the appropriate number using the number keypad to select the function to be performed (e.g., 2 will disable the automatic unload at the end of the job).

7. Press Enter
to accept the selection. The system will return to the ATC menu.

ATC Menu

Gang Drill Calibration

The Automatic Tool Changer (ATC) requires that the drills be calibrated only when a new drill bit is inserted into a drill in the holder. The machine will store the Z-axis tool offsets of the new tool, and this allows for fast tool changes. The Z-axis tool offset of each tool is stored in memory even after the machine is turned off.
Gang Drill Calibration can be completed after the operator has completed loading drills.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “A” in ATC.

3. Press Enter to access the ATC sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “C” in CalGDrill.

5. Press Enter to calibrate a gang drill. The #1 drill will move over to the calibration block or surface block.

6. Press Z-axis Jog Arrows

to lower the spindle until it is near the top of the block.

a. Press One

for “Manual” when calibrating only 1 drill after a bit change. When the drill has been calibrated, the display will return to the ATC menu.

b. Press Two

for “Auto” to begin the tool calibration. The Z-axis will move down slowly until it senses the block and will then lift off the block up to maximum Z height. Each drill will move over the block and go through the same motion until all are calibrated. When all of the drills are calibrated, the display will return to the ATC menu.

ATC Menu

AdjGDrill

The Adjust Gang Drill function allows the operator to select an individual drill and make minor adjustments. This prevents the operator from having to redo the settings on all drills when only one drill needs to be modified.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “A” in ATC.

3. Press Enter
to access the ATC sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “A” in AdjGDrill.

5. Press Enter
to access the gang drill settings individually.

6. Enter the number of the drill to be adjusted (e.g., 1) and press Enter .

7. Press X-axis Jog Arrows

to adjust the drill by the thousandths. Operators can make positive or negative adjustments.

8. Press Enter
to save the adjustment and return to the ATC sub-menu.

ATC Menu

Load Tools

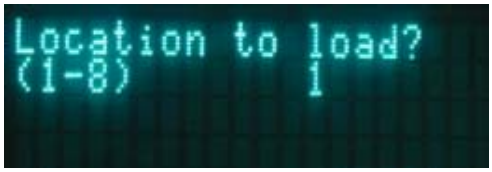
The Load Tools function is used to rotate the carousel on ATC systems. The current tool number will be displayed once LoadTools is selected.

1. Press Menu

to access the menu system.
2. Press Jog Arrows

until the blinking cursor is over the “A” in ATC.
3. Press Enter to access the ATC sub-menu.
4. Press Jog Arrows

until the blinking cursor is over the “L” in LoadTools.
5. Press Enter to access the Load Tools function.



6. Enter the location number of the tool to be loaded (e.g., 1).
7. Press Enter . The carousel will rotate to that tool number location.
8. Load the tool into the corresponding slot for that number.
9. Repeat steps 6-8 for the remaining tools.
10. Press Cancel to rotate the carousel back to its original position.

ATC Menu

Adjust Tool

The Adjust Tool function is used to adjust the length of a tool after the tool has been calibrated and the surface has been set. **Tools must be calibrated before they can be adjusted**.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

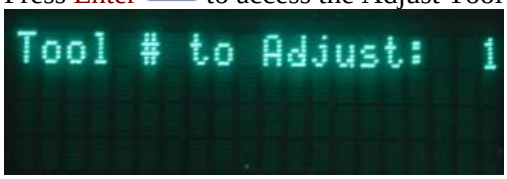
until the blinking cursor is over the “A” in ATC.

3. Press Enter to access the ATC sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “A” in Adj_Tool.

5. Press Enter to access the Adjust Tool function.



6. Enter the number of the tool to adjust using the number keypad (e.g., 1).

7. Press Enter to adjust the tool.

8. Enter the adjusting factor for the selected tool using the number keypad (e.g., -0.001). Positive numbers will increase and negative numbers will decrease the length of the tool.

9. Press Enter to adjust the tool. The display will return to the ATC menu. ATC Menu

Get_Tool

The Get_Tool function allows the operator to retrieve a specific tool and have it brought to the front of the machine.

1. Press Menu

to access the menu system.
2. Press Jog Arrows

until the blinking cursor is over “A” in ATC.
3. Press Enter to access the ATC sub-menu.
4. Press Jog Arrows

until the blinking cursor is over the “G” in Get_Tool.
5. Press Enter to access the Get_Tool function and enter the tool number (e.g., 1).

6. Press Enter to enable the spindle to retrieve the tool and bring it to the front of the machine.

ATC Menu

Load Drill

The Gang Drill function is used to allow easy access for operators to load and unload drills. The menu item should appear in the ATC sub-menu as LoadDrill.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over “A” in ATC.

3. Press Enter to access the ATC sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “L” in LoadDrill.

5. Press Enter to load a drill. The machine will move to the Hard Home position.

6. Enter the number of the drill to be lowered (e.g., 1) or press Set Max Depth to lower all drills.

7. Press numbers 2 through 9 to access each following drill. Any currently lowered drill is raised before the selected drill is lowered, and the lowered drill is displayed on the keypad. Operators can press Zero to retract any drill that is already lowered.

8. Adjust the set screw so that any existing bit can be removed or a new bit can be inserted.

9. Insert a drill bit into the drill.

10. Adjust the set screw to hold the bit securely.

11. Repeat steps 7-10 for each drill as needed.

12. Press Cancel to retract any lowered drill and return the display to the ATC sub-menu.

ATC Menu

CfgGDrill

The Configure Gang Drill function is used to set the machine to either unload the tool automatically or to establish an unload distance for manually unloading the tool**.**

1. Press Menu

to access the menu system.
2. Press Jog Arrows

until the blinking cursor is over “A” in ATC.
3. Press Enter to access the ATC sub-menu.
4. Press Jog Arrows

until the blinking cursor is over the “C” in CfgGDrill.
5. Press Enter to access the gang drill configuration.
6. Press Enter to toggle between No (N) and Yes (Y). Operators can set the unload distance if the configuration is set to No.

7. Press Cancel to return to the ATC sub-menu.

ATC Menu

Knife

The Knife sub-menu is used to establish the knife parameters specific to a routing machine. Operators may elect to order a tangential knife, tangential-oscillating knife, or cartridge knife. The tangential knife is designed to remain tangent to the material or profile during the cutting sequence, while the oscillating knife includes an up-and-down motion during the cutting. The cartridge knife allows the operator to switch out knives and access the saved parameters for each cartridge.

The tangential knife can be specialized as either a straight blade, a pizza-wheel blade that has a 45mm diameter and holder, or a creasing wheel that bends straight lines into material. The oscillating knife has a straight blade formation that moves at approximately 2000 Hz and is preferable for harder materials, excluding wood or metal. The cartridge knife offers up to 20 options for blade types. Most operators using the knife option are cutting vinyl, cloth, or plastic.

1. Press Menu

to access the menu system.
2. Press Jog Arrows

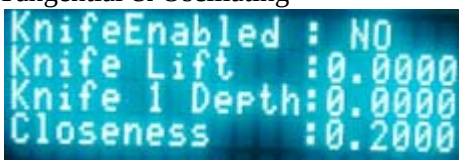
until the blinking cursor is over the “K” in Knife.
3. Press Enter to access the Knife sub-menu.
4. Press Jog Arrows

until the blinking cursor is over the appropriate option. Certain menu items will then enter a sub-menu with more selections.

Cartridge

Set Home w/Knife
Knife Control
XY Feedrate
ZDn Feedrate
Knife Lift
Knife Depth
Knife Dn Dly
Knife Up Dly
Closeness
Rotate Degree
Min. Radius

Tangential & Oscillating



KnifeEnabled
Knife Lift
Knife Depth
Closeness
Rotate Degree
Min. Radius
Home Knife
Set Knife Offset
Knife Test
Knife Slider Test
Set Knife Res

Menu

Set Home w/Knife

The Set Home w/Knife function is mainly used in the cartridge knife system when operators are not including the spindle in the job file. The setting can be turned ON and OFF.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in Knife.

3. Press Enter
to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “S” in Set Home w/Knife.

5. Press Enter
to set Home with the Knife.

6. Press Jog Arrows

until the knife is at the Home location and press Enter .

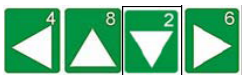
Knife Menu

Knife Control

The Knife Control function specifies whether the Knife Lift and Knife Depth will be set by the job file or at the keypad. Knife Lift and Knife Depth menu items are only available in the Knife sub-menu when the Knife Control is set to Keypad.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Enter to toggle between Job and Keypad.

Knife Menu

XY Feedrate

The Knife Feedrate function sets the cartridge knife cutting speed during the cutting sequence.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

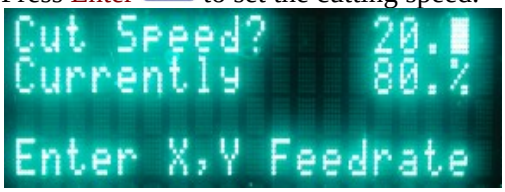
until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “X” in XY Feedrate.

5. Press Enter to set the cutting speed.



6. Input the cutting speed (e.g., 25) and press Enter . The value will be displayed on the Knife menu.

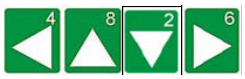
Knife Menu

ZDn Feedrate

The ZDn Feedrate function sets how fast the blade enters the material and is only applicable if the cartridge configuration is set to Z controlled.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “K” in Knife.

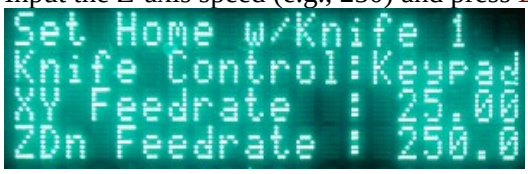
3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “Z” in ZDn Feedrate.

5. Press Enter to set the Z-axis speed.

6. Input the Z-axis speed (e.g., 250) and press Enter . The value will be displayed on the Knife menu.



Knife Menu

Knife Enabled

The Knife Enabled function makes any cutting machine a tangential or oscillating knife-only machine so that no other options, offsets, or cutting tools are recognized during regular operation. The operator should set a Soft Home from the center line of the knife rather than the center of the spindle. All M- and G-Code files that are run while the knife is enabled will operate corresponding standard tool files through the knife (e.g., files calling out Tool 1 will be run as Tool 61).

1. Press Menu

to access the menu system.

2. Press Jog Arrows

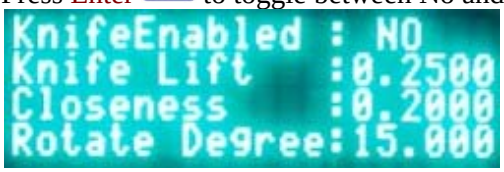
until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “K” in Knife Enabled.

5. Press Enter to toggle between No and Yes.



Knife Menu

Knife Lift

The Knife Lift function moves the knife above the material surface by a specified distance to rotate to a new degree and/or move to the next cut contour. With Tangential or Oscillating knives, the Knife Lift is not affected by the tool lift set in the M- and G-Code files, and the operator must set the Knife Lift value from the Knife menu options before executing the cut file. With Cartridge knives, the Knife Lift option is only available if the Z Control configuration is set up as Keypad; otherwise, the Z values from the job file are used.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “K” in Knife Lift.

5. Press Enter to set the Knife Lift.

6. Enter lift height (e.g., 1.500) and press Enter . The value will be displayed on the Knife menu.

Knife Menu

Knife Depth

The Knife Depth function refers to the Z screw distance that the knife will move down from the surface of the material during the file execution. With Tangential or Oscillating knives, Knife Depth is not related to the oscillation stroke, not affected by the Cut Depth, and not controlled by M- and G-Code files, and the operator must set the Knife Depth value from the Knife menu for the proper depth of cut in the file execution. With Cartridge knives, the Knife Depth menu option is only available if the Z Control configuration is set up as Keypad; otherwise, the Z values from the job file are used.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “K” in Knife Depth.

5. Press Enter to set the Knife Depth.

6. Enter a cut depth (e.g., 0.0) and press Enter . The value will be displayed on the Knife menu.

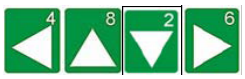
Knife Menu

Knife Dn Delay

The Knife Dn Delay function is only used when a cartridge is configured as Solenoid controlled. When the cartridge is lowered, the Knife Dn Delay allows the system to wait for the cartridge to lower into position. If the cartridge is Z controlled, then no delay will occur.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “K” in Knife Dn Delay.

5. Press Enter to set the Knife Dn Delay.

6. Enter the value (e.g., 1000msec) and press Enter . The value will be displayed on the Knife menu.

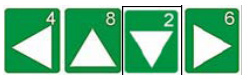
Knife Menu

Knife Up Delay

The Knife Up Delay function is only used when a cartridge is configured as Solenoid controlled. When the cartridge is raised, the Knife Up Delay allows the system to wait for the cartridge to raise into position. If the cartridge is Z controlled, then no delay will occur.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “K” in Knife Up Delay.

5. Press Enter to set the Knife Up Delay.

6. Enter the value (e.g., 1000msec) and press Enter . The value will be displayed on the Knife menu.

Knife Menu

Closeness

The Closeness function is used to set how closely the knife will follow the job file. The Closeness value is calculated from the vertex point inward between the two straight lines. Although the input value is the same for all the varying degrees (e.g., 0.010 closeness on angles of 22°, 95°, and 154°), it affects those angles differently. The more obtuse or dull the angle, the more effect the closeness value of 0.010 has. The more acute or sharp the angle, the less effect the closeness value has.

- Operators who wish to **enable rounding** should set a low closeness value and a high minimum radius value.
- Operators who wish to **disable rounding** should set a high closeness value and a low minimum radius value.

If the arc from the closeness value is less than the minimum radius value, then the knife will move to the vertex, lift up to the Knife Lift Height, rotate to follow the path exactly, and re-enter the material to continue the job file.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

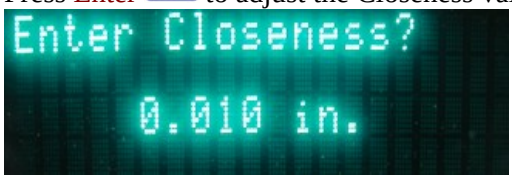
until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “C” in Closeness.

5. Press Enter to adjust the Closeness value.



6. Enter the closeness value using the number keypad (e.g., 0.010).

7. Press Enter to save the value and return to the Knife menu where the Closeness value will be displayed.

Knife Menu

Rotate Degree

The Rotate Degree function sets the degree the blade can stay down in the material and rotate if the blade is at the vertex point of an angle. **Rotate Degree is only used when the closeness arc created is smaller than the minimum radius**; therefore, this parameter is only used when the closeness value is very low (e.g., 0.010) and the minimum radius is very high (e.g., 0.750).

The controller will move the knife to the vertex point and then compare the relative angle of the required theta motion to the rotate degree value input at the keypad. With the knife tip at the vertex angle, the knife will twist to the new angle while still in the material and then continue in X & Y. Some operators will use a lower degree for thicker materials and a higher degree for thinner materials to prevent extra indentations in the material from where the knife rotated.

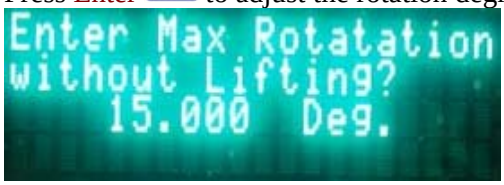
- Operators who wish to **enable rounding** should set a low closeness value, a high minimum radius value, and a rotate degree value of greater than 90.
- Operators who wish to **disable rounding** should set a high closeness value, a low minimum radius value, and a rotate degree value of 90.

1. Press Menu

to access the menu system.
2. Press Jog Arrows

until the blinking cursor is over the “K” in Knife.
3. Press Enter to access the Knife sub-menu.
4. Press Jog Arrows

until the blinking cursor is over the “R” in Rotate Degree.
5. Press Enter to adjust the rotation degree.



6. Enter the maximum rotation degree using the number keypad (e.g., 15.000).
7. Press Enter to save the value and return to the Knife menu where the Rotate Degree value will be displayed.

Knife Menu

Min. Radius

The Minimum Radius function establishes the smallest radius that the knife will cut and is only applicable at 2 connecting line entities. If the arc created by the closeness value is larger than the minimum radius value, then the knife will follow the closeness value in the X- and Y-axes. This effectively keeps the knife in the material for a smooth motion.

If the arc that is automatically created by the closeness value is smaller than the minimum radius value, then the knife will move to the vertex point, lift, rotate, lower back down, and continue cutting. The parameters at the keypad have little or no effect on arc commands in G Code.

- Operators who wish to **enable rounding** should set a high minimum radius value and a low closeness value.
- Operators who wish to **disable rounding** should set a low minimum radius value and a high closeness value.

1. Press Menu

to access the menu system.
2. Press Jog Arrows

until the blinking cursor is over the “K” in Knife.
3. Press Enter to access the Knife sub-menu.
4. Press Jog Arrows

until the blinking cursor is over the “M” in Min. Radius.
5. Press Enter to access the minimum radius setting.

6. Enter a value for the minimum radius using the number keypad (e.g., 0.750).
7. Press Enter to save the value and return to the Knife menu where the Minimum Radius value will be displayed.

Knife Menu

Home Knife

The Home Knife function is used to set the Home location for the tangential, oscillating, or cartridge knife, which is on a separate stepper motor than the spindle. Operators will not have to re-home the knife unless the Flash parameter changes or the knife has lost the last physical Home location.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in Knife. Operators of Cartridge Knife systems would select the “K” in KnifeUtil.

3. Press Enter to access the Knife sub-menu. Operators of Cartridge Knife systems would access the KnifeUtil sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “H” in Home Knife.

5. Press Enter to access the Home Knife function.

6. Wait until the Knife has located its Home position and the keypad shows the Knife menu before continuing with the setup or cut sequence. Operators of Cartridge Knife systems would wait for the KnifeUtil menu to show on the keypad before continuing.

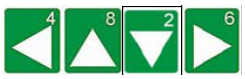
Knife Menu

Set Knife Offset

The Set Knife Offset function is used at the initial setup to set the orientation of the tangential or oscillating knife parallel to the X-axis. This configuration will usually be completed when the machine is installed and should only be adjusted if the knife shifts out of alignment with the X-axis.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “K” in K Offset.

5. Press Enter to adjust the knife offset.

6. Enter the number for the knife to be adjusted (e.g., 3) and press Enter to accept the value.

7. Press Jog Arrows

to reach the Knife Offset and press Enter .

8. Press Enter to accept the change. The knife will move to the offset, and the display will return to the Knife menu.

Knife Menu

Knife Test

The Knife Test function is used to test the responsiveness of the tangential, oscillating, or oscillating knife before beginning the cutting sequence. This process can be repeated if the knife is changed or the current knife is having difficulty completing the cutting sequences.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

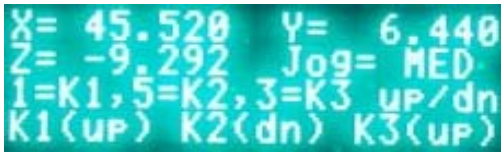
until the blinking cursor is over the “K” in Knife. Operators of Cartridge Knife systems would select the “K” in KnifeUtil.

3. Press Enter to access the Knife sub-menu. Operators of Cartridge Knife systems would access the KnifeUtil sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “K” in Knife Test.

5. Press Enter to begin the knife test.



or

6. Enter the number for the knife to be tested (e.g., 1=K1) and press Enter . The knife will move down along the Z-axis.

7. Press the same number again to move the knife back up. The screen will show *up* or *dn* to indicate the position of the knife.

8. Press Cancel to exit the test and return to the Knife menu. Operators of Cartridge Knife systems will return to the KnifeUtil menu.

Knife Menu

Knife Slider Test

The Knife Slider Test function is used to test the responsiveness of the entire knife assembly before beginning the cutting sequence. This process can be repeated if the tangential, oscillating, or cartridge knife is changed or the current knife is having difficulty completing the cutting sequences.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in Knife. Operators of Cartridge Knife systems would select the “K” in KnifeUtil.

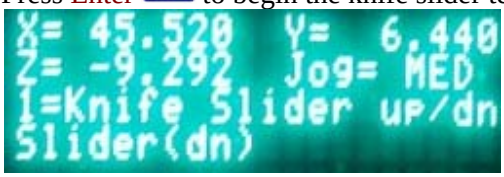
3. Press Enter to access the Knife sub-menu. Operators of Cartridge Knife systems would access the KnifeUtil sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “K” in Knife Slider Test.

5. Enter the knife number if the machine is equipped with the Cartridge Knife system and press Enter .

6. Press Enter to begin the knife slider test.



7. Enter 1 to move the knife slider along the Z-axis.

8. Press 1 again to move the knife slider back up. The screen will show *up* or *dn* to indicate the position of the knife slider.

9. Press Cancel to exit the test and return to the Knife menu. Operators of Cartridge Knife systems will return to the KnifeUtil menu.

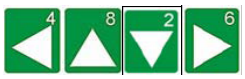
Knife Menu

Set Knife Res

The Set Knife Resolution function is used to modify the orientation of the tangential or oscillating knife. This process is usually completed at setup and will not have to be readjusted unless the knife is replaced or shifted out of alignment.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

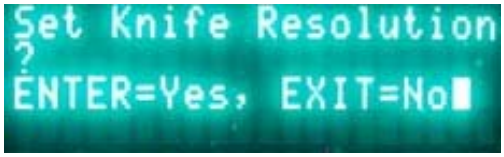
until the blinking cursor is over the “K” in Knife.

3. Press Enter to access the Knife sub-menu.

4. Press Jog Arrows

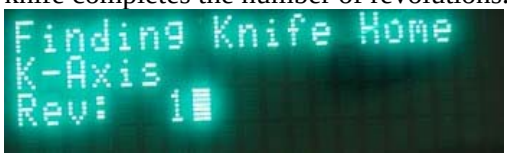
until the blinking cursor is over the “K” in Knife Resolution.

5. Press Enter to access the Knife Resolution function.



6. Press Enter to set the knife resolution.

7. Enter the number of revolutions using the number keypad (e.g., 10). The display will count down while the knife completes the number of revolutions.



8. Review the Knife Resolution change on the display.

9. Press Enter to find the Knife Home and return to the Knife menu.

Knife Menu

Innovation. Quality. Performance. Best in Industry. Worldwide.

KnifeUtil

The Knife Utility sub-menu contains menu items for the Cartridge Knife that are usually set once and never have to be changed.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in KnifeUtil.

3. Press Enter to access the Knife Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the appropriate option. Certain menu items will then enter a sub-menu with more selections.

Cartridge #
Cartridge Config.
Home Knife
Osc Stroke
Cartridge Offsets
Move to Knife Offset
Knife Test
Knife Slider Test
Knife Osc Test
Set Knife 1 Res
Set Knife 2 Res

Menu

Innovation. Quality. Performance. Best in Industry. Worldwide.

Cartridge Number

The Cartridge # function rotates the cartridge receiver to that cartridge number's offset for visual verification. The system will default to Cartridge 31 at initial boot-up and during regular operations if no cartridge is already set up in the machine.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the "K" in KnifeUtil.

3. Press Enter
to access the Knife Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the "C" in Cartridge #.

5. Press Enter
to access the cartridge.

6. Input the cartridge number and press Enter
. The display returns to the KnifeUtil menu showing the selected cartridge.

KnifeUtil Menu

Cartridge Config

The Cartridge Config function determines how each knife operates in a job. Each knife can be configured as drag or oscillating, and the oscillations will become active after the knife is selected. Additionally, each knife can be Z controlled, Solenoid controlled, or Angled. Operators **must** resurface and recalibrate the cartridge knife after changing configurations.

- If Z controlled, the Cartridge Configuration function uses the Knife Depth and Lift values from the executed M- & G-Code file. Z Control is the preferred method for cutting material that is 1" (25.4mm) or thicker.
- If Solenoid controlled, the Cartridge Configuration function uses the Knife Up Dly and Knife Dn Dly in addition to raising and lowering the knife and is the preferred method for cutting material that is 3/4" (19.1mm) or thinner.
- If Angled, the Cartridge Configuration function accounts for the 45° angle of the blade

1. Press Menu

to access the menu system.

2. Press Jog Arrows

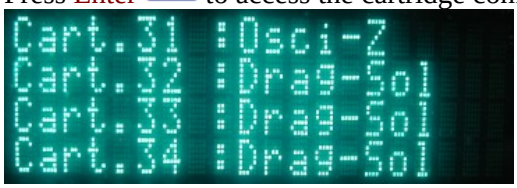
until the blinking cursor is over the "K" in KnifeUtil.

3. Press Enter to access the Knife Utility sub-menu.

4. Press Jog Arrows

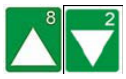
until the blinking cursor is over the "C" in Cartridge Config.

5. Press Enter to access the cartridge configuration list.



6. Press X-axis Jog Arrows

to reach the applicable cartridge number and press Enter .

7. Press X-axis Jog Arrows  to reach the appropriate parameter and Y-axis Jog Arrows to change the setting. The asterisk (*) indicates which setting is active.

   If the cartridge is configured as Solenoid rather than Z up/dn, then the Angle/Straight option is no longer available.

8. Press Enter to return to the Cartridge Configuration sub-menu or Cancel to return to the KnifeUtil menu.

KnifeUtil Menu

Osc Stroke

The Osc Stroke function shows the value that will be added to the length of the tool during the surfacing or calibrating of the knife when the knife is not oscillating. This setting is only valid on an oscillating cartridge with the oscillations turned off during setting surface or calibrating. The variable stroke is based on the PSI set by the regulator.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in KnifeUtil.

3. Press Enter to access the Knife Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “O” in Osc Stroke.

5. Press Enter to set the value of the oscillation stroke.

6. Input a value (e.g., 0.340) and press Enter . The display will return to the KnifeUtil menu where the Oscillating Stroke will be displayed.

KnifeUtil Menu

Cartridge Offsets

The Cartridge Offsets function is the theta offset for each cartridge. Cartridge 31 is the limit offset for knife head 1 after homing, and cartridge 41 is the limit offset for knife head 2 after homing. The cartridge offsets 32 – 40 are relative to cartridge offset 31. Operators may set the cartridge offset by Jogging the theta using the Z keys until the knife cartridge faces X positive. Once the offset is located, operators should save the offset for that cartridge. The knife should face X positive after homing.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in KnifeUtil.

3. Press Enter to access the Knife Utility sub-menu.

4. Press Enter to access the Cartridge Offsets.

5. Press X-axis Jog Arrows

to reach the specific cartridge and press Enter . The machine will find the knife home and then issue the following prompt:

6. Press Jog Arrows

to Jog to the knife offset and press Enter . The display returns to the Cartridge Offsets screen.

KnifeUtil Menu

Move to Knife Offset

The Move to Knife Offset function is a relative move from the current position and is used to validate the correct setting for the knife offsets.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in KnifeUtil.

3. Press Enter to access the Knife Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “M” in Move to Knife Offset.

5. Press Enter to access the Move to Knife Offset function.

6. Enter the knife number (e.g., 1 or 2) and press Enter to move to the offset. The display will return to the KnifeUtil menu.

KnifeUtil Menu

Knife Osc Test

The Knife Osc Test checks the Knife Oscillator output.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in KnifeUtil.

3. Press Enter to access the Knife Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “K” in Knife Osc Test.

5. Press Enter to access the oscillation test.

6. Enter the knife number (e.g., 1 or 2) and press Enter .

7. Press 1

to toggle the oscillation test ON and OFF.

8. Press Cancel to exit the oscillation test. The display will return to the KnifeUtil menu.

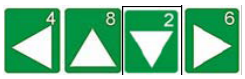
KnifeUtil Menu

Set Knife 1 Res

The Set Knife 1 Res function determines the knife resolution for the first knife if the technician does not know the gearing. The Set Knife 1 Resolution function uses the knife limit switch to measure the distance required to complete 1 revolution.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “K” in KnifeUtil.

3. Press Enter to access the Knife Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “S” in Set Knife 1 Res.

5. Press Enter to access the Set Knife 1 Resolution function.

6. Press Enter to set the resolution.

7. Enter the number of revolutions and press Enter . The knife will complete the input number of revolutions, count down on the display, and display any changes to Knife 1 resolution.

8. Press Enter to save the new resolution and return to the KnifeUtil menu. KnifeUtil Menu

Set Knife 2 Res

The Set Knife 2 Res function determines the knife resolution for the second knife if the technician does not know the gearing. The Set Knife 2 Resolution function uses the knife limit switch to measure the distance required to complete 1 revolution. This option will not be available on single-knife systems.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

until the blinking cursor is over the “K” in KnifeUtil.

3. Press Enter to access the Knife Utility sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “S” in Set Knife 2 Res.

5. Press Enter to access the Set Knife 2 Resolution function.

6. Press Enter to set the resolution.

7. Enter the number of revolutions and press Enter . The knife will complete the input number of revolutions, count down on the display, and display any changes to Knife 2 resolution.

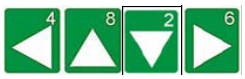
8. Press Enter to save the new resolution and return to the KnifeUtil menu. KnifeUtil Menu

Material Handling

The Material Handling sub-menu is used to operate the material handling and material pushing option on the 5000 Series XD line of machines.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “M” in MatlHand.

3. Press Enter to access the Material Handling sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the appropriate option. Certain menu items will then enter a sub-menu with more selections.

Roller Setup …
Material Pusher …
Material Lifter …
Air/Knife Sweeper
PopUp Pins

Menu

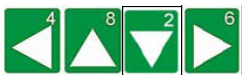
Innovation. Quality. Performance. Best in Industry. Worldwide.

Roller Setup

The Roller Setup sub-menu is used to operate the rollers used in holding down multiple sheets of material. The rollers can be operated in sequence and can be tested individually.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

until the blinking cursor is over the “M” in MatlHand.

3. Press Enter to access the Material Handling sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “R” in Roller Setup.

5. Press Enter to access the Roller Setup.

6. Press X-axis Jog Arrows

to select the appropriate option.

- Work Start will set the coordinate location for the initial cutting sequence.
- Sheet Size will configure the system for the size of the material.
- Enable Rollers will allow the rollers to be activated once the material is in the proper place.
- Pre-Lift will allow a certain amount of space before lifting the material.
- Roller 1 Position will show the coordinate location of roller 1 relative to the spindle.
- Roller 2 Position will show the coordinate location of roller 2 relative to the spindle.
- Roller 3 Position will show the coordinate location of roller 3 relative to the spindle.
- Roller 4 Position will show the coordinate location of roller 4 relative to the spindle.
- Test I/O will complete the internal connection to determine communication.

7. Press Cancel to return to the Material Handling sub-menu.

Material Menu

Material Pusher

The Material Pusher sub-menu is used to move the material into place for cutting and off the table once the cutting sequence has been completed. For safety reasons, the Material Pusher will only fire down over the work and unload areas of the table

1. Press Menu

to access the menu system.

2. Press Jog Arrows

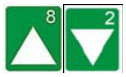
until the blinking cursor is over the “M” in MatlHand.

3. Press Enter to access the Material Handling sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “M” in Material Pusher.

5. Press Enter to access the Material Pusher sub-menu.

6. Press X-axis Jog Arrows 

to select the appropriate option.

- Set Start Position will set the beginning position for the material pusher.
- Set Stop Position will set the ending position for the material pusher.
- Set Push Feedrate will set the speed at which the pusher will operate.
- Auto Unload will unload the material automatically.
- Test Cycle will complete a sequence to determine response.
- Test I/O will complete the internal connection to determine communication.

7. Press Cancel to return to the Material Handling sub-menu.

Material Menu

Material Lifter

The Material Lifter sub-menu is used to create a suction on the material, lift the material from the stack, and raise the material to the proper height before lowering it to the cutting table.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

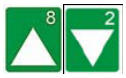
until the blinking cursor is over the “M” in MatlHand.

3. Press Enter to access the Material Handling sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “M” in Material Lifter.

5. Press Enter to access the Material Lifter sub-menu.

6. Press X-axis Jog Arrows 

to select the appropriate option.

- Num Sheets will indicate the number of sheets of material with a maximum of 5 sheets.
- Auto Load will set the configuration to load at the start of a job or at the sheet.
- Load Prompt will set the configuration to prompt at the start of a job or at the sheet.
- Lift Delay will set the time lapse before the material lifter will raise.
- X Pickup will specify the X-axis location at the back to pick up the material.
- X Drop-off will specify the X-axis location to drop off the material for final adjustments.
- Load Speed will set the speed at which the material should be loaded.
- XPos Dist will indicate the X position for laying the material onto the table.
- XPos Speed will indicate the feedrate at which the material will be moved onto the table.
- Test Cycle will complete a sequence to determine response.
- Put Sheet Away will place the sheet of material out of the cutting area.
- Test I/O will complete the internal connection to determine communication.

7. Press Cancel to return to the Material Handling sub-menu.

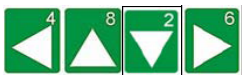
Material Menu

Air/Knife Sweeper

The Air/Knife Sweeper sub-menu is used to test the responsiveness of the air knife by blowing a high-pressure airflow onto the table between the front roller and the sweeper.

1. Press Menu

to access the menu system.

2. Press Jog Arrows 

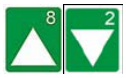
until the blinking cursor is over the “M” in MatlHand.

3. Press Enter to access the Material Handling sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “A” in Air/Knife Sweeper.

5. Press Enter to access the Air/Knife Sweeper sub-menu.

6. Press X-axis Jog Arrows 

to select the appropriate option.

- Sweeper Delay will impose a set period of time in microseconds before operating the air knife sweeper.
- Test I/O will complete the internal connection to determine communication.

7. Press Cancel to return to the Material Handling sub-menu.

Material Menu

PopUp Pins

The PopUp Pins sub-menu is used to test the responsiveness of the popup pins and the level of connectivity.

1. Press Menu

to access the menu system.

2. Press Jog Arrows

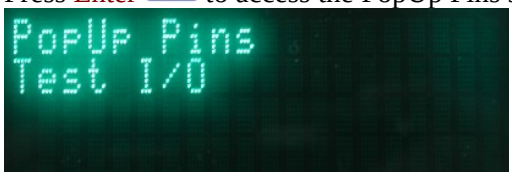
until the blinking cursor is over the “M” in MatlHand.

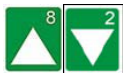
3. Press Enter to access the Material Handling sub-menu.

4. Press Jog Arrows

until the blinking cursor is over the “P” in PopUp Pins.

5. Press Enter to access the PopUp Pins sub-menu.



6. Press X-axis Jog Arrows 

to select the appropriate option.

- Test I/O will complete the internal connection to determine communication.

7. Press Cancel to return to the Material Handling sub-menu.

Material Menu

Options

top

Chip Removal Options 2
- Pressure Foot 2
- Dust Collection Hoods 3
- Dust Collection Hoods Installation for the Dust Boot 5

Engraver 6

Pneumatic Drill 6

Vinyl Knife 7

Laser Pointer 8

Table Format 9
- Vacuum Options 10
- Vacuum Zones 11
- Locator Pins 12

MD1200 Micro Drop Misting System 14
- Precautions for the MD1200 Micro Drop Misting System 14
- Operating the MD1200 Micro Drop Misting System 15
- Troubleshooting the MD1200 Micro Drop Misting System 16

Coolubricator System 17
- Operating the Coolubricator System 18
- Troubleshooting the Coolubricator System 19

Chip Removal Options

Chip removal with any routing system is vital to machine performance, tool life, and the improvement of edge quality. ***MultiCam*** has developed different options to achieve chip removal needs in the routing industry. These options include the Pressure Foot and the Dust Collector, both of which remove unwanted waste material during and after a project.

Options

Pressure Foot

The Pressure Foot can be used for both removing waste during the cutting operation and applying pressure to the surface of the material to reduce vibration and keep parts from moving. Its innovative design applies adjustable spring pressure, which makes the Pressure Foot an ideal option for cutting paper-masked plastics and acrylic materials. Equipped with a standard 2” hose input, the Pressure Foot is typically used with a high pressure vacuum and can lift with the spindle to avoid hitting clamps or other obstructions.

1. Use glide pad attachments when the material needs to be held down. Glide pad attachments are not recommended for materials with a fine finish. Brush attachments are a better choice for materials that scratch easily.

2. Adjust the height of the Pressure Foot so that the bottom of the glide pad is approximately 1/8” below the bottom of the routing bit when the spindle lifts.

3. Turn the wing nut on the height adjustment shaft to move the pressure foot up and down.

4. Re-tighten the wing nut once it is at the correct height.

5. Adjust the springs so that they are compressed halfway when the bit is at full depth in the material.

Options

Dust Collection Hoods

MultiCam can also provide a high flow dust collection hood for most of the collet and ATC spindles. The dust collector attachment is designed to be used with high CFM dust collection systems generally found in wood shops.

Dust Boot Dust Collector

The Dust Boot is designed to be used with a collet spindle in high CFM dust collection environments. With a 4" hose input, the dust boot moves with the Z-axis.

Pull-up Dust Collector for ATC Spindle

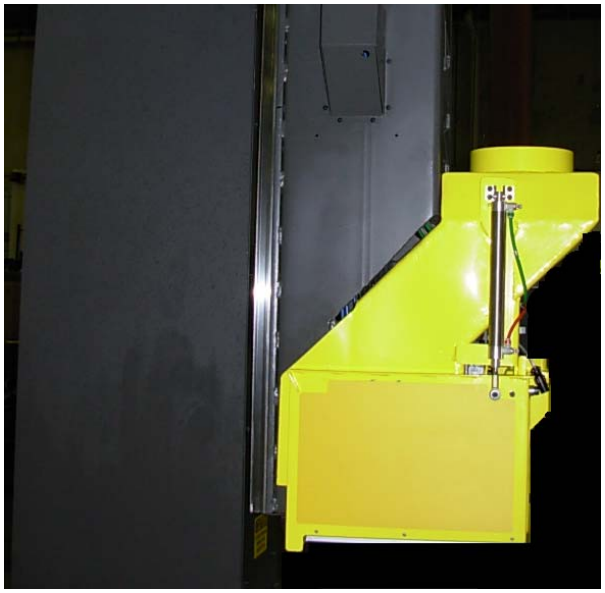
The Pull-up Dust Collector is designed for use in a high CFM dust collection environment. With a 4" hose input, the pull-up dust collector lifts when the spindle is not running and moves with the Z-axis.

Dust Collector for ATC Spindle

The Dust Collector for the ATC Spindle is designed for use in a high CFM dust collection environment. With a 4” hose input, the dust collector lifts when the spindle is not running and moves with the Z-axis.

Pantleg Dust Collector

The Pantleg Dust Collector is designed to be used with machines that access cutting sequences on the C-axis with CFM dust collection systems. More specialized than other dust collectors, the Pantleg Dust Collector is equipped with an 8” input hose and the capability for maneuvering very large tools. The dust collector lifts up when the spindle stops moving, and all movement follows the Z-axis.



Options

Dust Collection Hoods Installation for the Dust Boot

Mounting brackets and clamps for the Dust Boot dust collection hood are located on both sides of the spindle. The operator must identify these prior to installing the dust collection hood.

1. Locate the pocket into which the spindle will fit on the top plate of the dust collection hood.

2. Place the dust collection hood under the spindle and lift it into the spindle.

3. Place the holding clamps over the mounting bracket and flip them into place.

4. Connect a 4” vacuum hose to the front of the hood. The standard dust collection hood has a 4” vacuum hose connection point, and adapters are available at any hardware store for proper connectivity if needed.

5. Provide enough hose for the dust collection hood to travel the entire length of the table. The hose should be suspended from above the table to prevent it from being pulled out or otherwise damaged as the system moves.

Options

Engraver

MultiCam offers an engraving spindle that is ideal for most engraving applications and only available with certain systems. The floating nose engraver is belt-driven directly from the router spindle, and the spindle accepts standard top-loading engraving cutters.

The engraving spindle is available with a standard 1/4” collet, though other collet sizes such as 11/64” are available.

Options

Pneumatic Drill

The Electric/Pneumatic Drill adds up to 2 additional drills to the ***MultiCam*** Router for additional production efficiency. The drill and spindle operate from the same job file, and no proprietary interface software is necessary to operate.

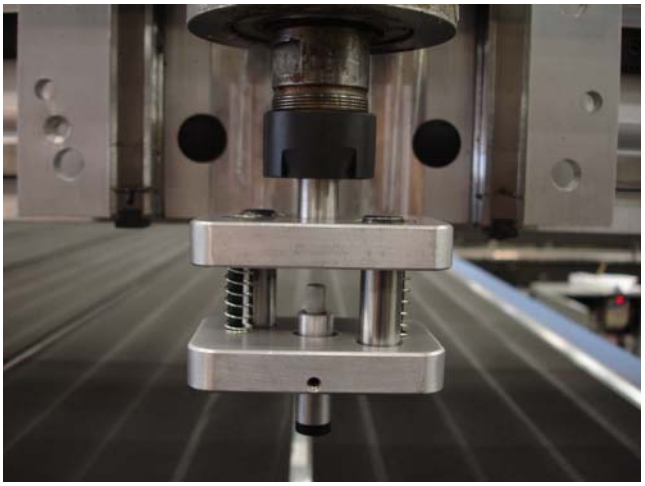
The Pneumatic Drill requires compressed air but operates at 3000 RPM on a standard 3/8” chuck.

Options

Vinyl Knife

The Vinyl Knife option is available for all routers to cut thin materials, such as vinyl labels, following the same cut file as the router file. The vinyl knife can be inserted into the chuck just like a regular tool and can also be inserted into the tool changer for ATC machines. Operators can run regular cut files as well as test files using the vinyl knife, which is spring-loaded for more accurate pressure on the material.

When configuring the tool, operators will need to set the tool to No Spin to get a better quality performance. Operators must manually set surface with the vinyl knife and must always use extreme caution when handling the blade.



Options

Innovation. Quality. Performance. Best in Industry. Worldwide.

Laser Pointer

The Laser Pointer option allows operators to set Home using the laser diode rather than relying solely on naked eye approximations. With the laser diode mounted on the carriage of the machine and pointed at the material surface, operators can see exactly where the cutting sequence will begin and set their Home locations exactly. Once the offset is established, operators can begin using the laser pointer to set Home. The offset will not need to be reset unless the laser diode is replaced or loses its orientation to the cutting head.

1. Turn on the Laser Pointer by pressing 0 . Operators can toggle the Laser Pointer ON and OFF* by pressing 0, and the Home screen will show whether or not the Laser Pointer is active by inserting the letter L in the lower right corner. (Operators who wish to set Home with the standard method should leave the Laser Pointer OFF.) The following examples show how the X and Y values change when the Laser Pointer is toggled ON or OFF.

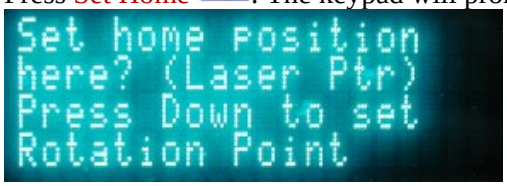
Laser Pointer ON

Laser Pointer OFF

* **Operators cannot set Home using the Laser Pointer if the Laser Pointer is OFF**.

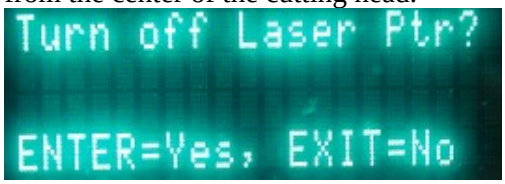
2. Jog the Laser Pointer to the Home location.

3. Press Set Home . The keypad will prompt the operator to set the Home location using the Laser Pointer.



4. Press Enter to set Home.

5. Press Enter to leave the Laser Pointer ON or Cancel to turn the Laser Pointer OFF. If the Laser Pointer is left on, the laser beam will **not** follow the cutting head as it cuts since the Laser Pointer is offset from the center of the cutting head.



6. Finish system preparations and run the file.

Options

Table Format

The table option is available in different formats for all routers. The T-plate table is the preferred format for bolting down parts that have irregular shapes or parts that cannot be held in place securely with a vacuum. The vacuum T-plate is the preferred format for bolting down parts that react well to the vacuum but may be irregularly shaped. The gridded vacuum is the preferred format for holding down parts that have large, flat surfaces that respond well to a vacuum system.

T-plate Table
and
Vacuum T-plate Table

Gridded Vacuum Table

Options

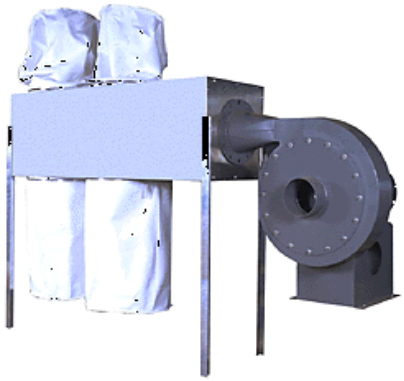
Vacuum Options

The vacuum pump option is available with different levels of horsepower. The sales associate can help operators determine which type of vacuum pump and in what configuration would be the most effective for the machine and the material being cut.

Positive Displacement (PD) vacuum pumps are available in 10hp, 20hp, and 30hp at 200 CFM, 425 CFM, and 700 CFM respectively. PD pumps are used mainly with nested and random part manufacturing.

High Pressure vacuum pumps are available in 5hp, 10hp, and 15hp at 70, 177, 238 CFM respectively. While all three of the high pressure vacuum pumps are used mainly with fixture parts, the 15hp high pressure vacuum pump is also used with random part manufacturing.

High Flow vacuum pumps are available in 25hp at 3000 CFM. High flow vacuum pumps are used mainly with large tables that hold large parts.



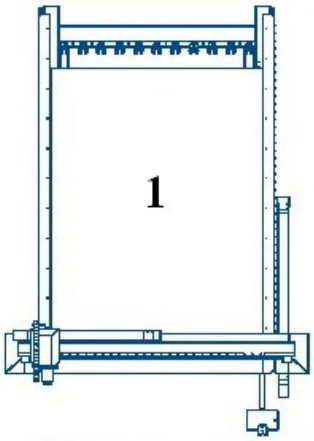
Regenerative Blowers are available in 8.5hp, 17hp, and 33hp at 215 CFM, 353 CFM, and 776 CFM. Regenerative blowers are used mainly with random part manufacturing.

Options

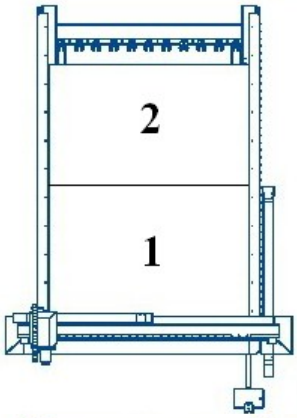
Vacuum Zones

The vacuum zone option is configured per table. Operators can select how the different parts of the table will receive the vacuum holding. The zones can be set up as 1, 2, 4, or custom.

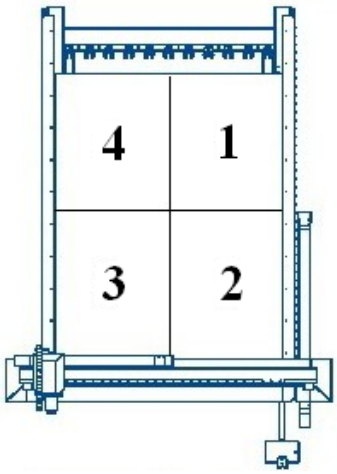
1 zone provides equal vacuum pressure to the entire table.



2 zone provides vacuum pressure to the top or bottom of the table.



4 zone provides vacuum pressure to the four corners of the table.

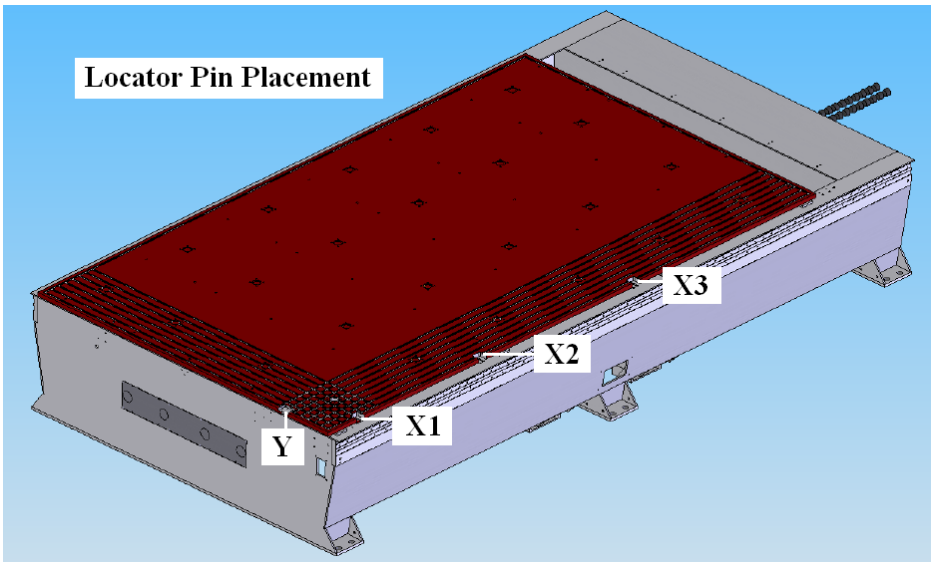


Custom zone can be modified per the operator's specifications.

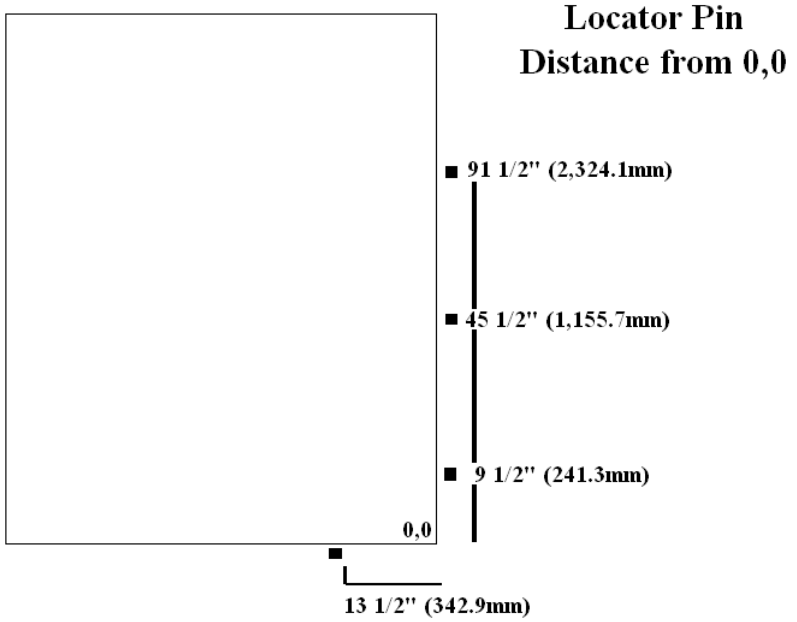
Options

Locator Pins

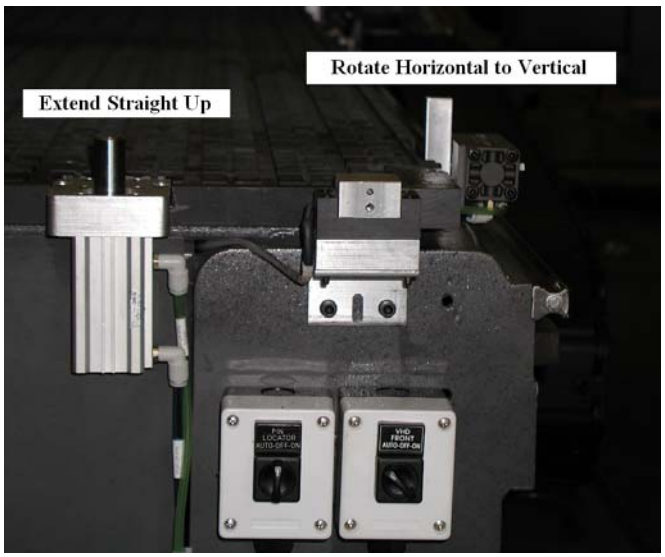
Operators have the option of requesting locator pins on their 3000 and 5000 Series Router machines. The locator pins allow the operator to maintain the material alignment at 0,0 when cutting features along the surface or through the material. Features can be guaranteed in the right place since the locator pins prevent any unintentional adjustments of the material during the cutting sequence.

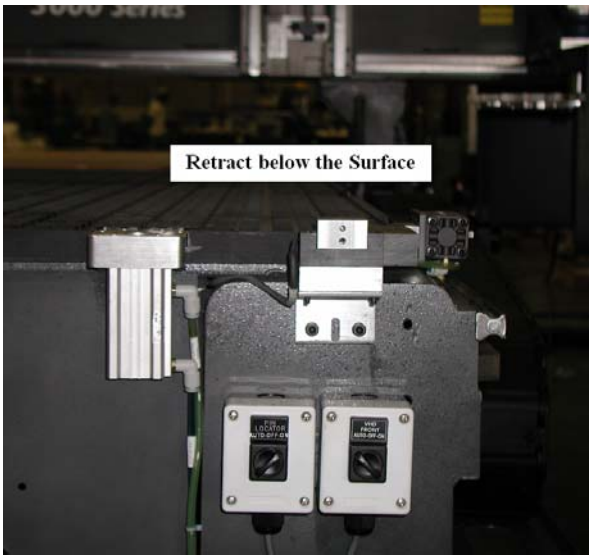


Routers with the standard locator pin option will have one locator pin along the Y-axis and two or three locator pins along the X-axis depending on the size of the table and the customer preference. Standard locator pin placement is Y, X1, and X3 with the operator's ability to move the X3 locator pin to X2. Customers can choose different locator pin locations as needed. The standard distances are shown below.



Locator pins can either extend straight up and down 2” (51mm) or more above the table surface or rotate 2” (51mm) or more horizontal to vertical along the X-axis. The Y-axis locator pin as well as standard locator pins moves straight up and down, but wider tables may require locator pins that rotate horizontal to vertical.





Options

MD1200 Micro Drop Misting System

The MD1200 Micro Drop Misting System was an option that connected to the back of the bridge and worked to reduce friction during the cutting sequence so that operators may get a good cut with metal. Unlike conventional flood or mist systems that simply remove cutting-tool heat, micro dispensing prevents damaging heat build-up by eliminating friction, the source of heat. This option is no longer offered on new machines.

The automatic anti-siphon valves prevent forward or reverse flow of lubricant when airflow is turned off with either the main power switch or the airflow adjustment, allowing operators to avoid waiting for the lubricant at start-up or cleaning up the overflow after powering down. The nozzles may be mounted in any position relative to the dispenser, and the bayonet mount reservoir allows for easy filling.

The MD1200 Micro Drop Misting System applies very small amounts of high performance lubricant, atomizing the lubricant in a high velocity air-stream and delivering the lubricant to the cutting tool. This approach results in high material removal rates, improved finish, tighter tolerances, increased productivity, and eliminated costs for coolant or lubricant maintenance and disposal. The simple high resolution controls make adjustments fast and easy, and the direct reading fluid scale can be calibrated in drops per minute. With no confusing timer to set, the Micro Drop Misting System offers maximum chip clearing power with no fog for the high energy nozzles.

Options

Precautions for the MD1200 Micro Drop Misting System

Please read all instructions before operating the MD1200 Micro Drop Misting System to avoid damage to personnel or the machine.

1. Use only Trico Micro Drop lubricants.

2. Avoid using airline lubricators, water-based lubricants, water-based coolants, and flammable oils.

3. Use only clean, dry shop air between 60 and 125 psi.

4. Use a forty (40) micron filtration.

5. Empty the system when transporting.

6. Avoid spraying flammable liquids as this may damage the machine.

Options

Operating the MD1200 Micro Drop Misting System

The MD1200 features individual air and liquid flow adjustments for each nozzle. The following guidelines should be observed when adjusting the MD1200.

1. Adjust the airflow setting to register between the MIN and MAX settings indicated on the front panel of the unit. Airflow settings below the MIN point will cause the anti-siphon valve to engage and cut off the flow of lubricant, while airflow settings above the MAX point may cause fog.

2. Adjust the airflow high enough to give good chip clearing power without exceeding the MAX setting on the scale.

3. Start with the liquid setting between 12 and 16 drops/min.

4. Adjust the liquid flow after the unit has been allowed to stabilize to steady operation. This should take 2 to 3 seconds.

5. Observe the cutting tool after a cut has been made. If excess lubricant is present, decrease the liquid flow setting. Ideally, all lubricant should be consumed during the cutting process.

System Specifications of the MD1200 Micro Drop Misting System

System type: Pressure type, non-cyclic Micro-dispenser
Fluid viscosity range: 50-200 SUS
Liquid flow rate range: 2-16 drops/min
Inlet pressure range: 60-125 psi
Recommended filtration: 40 micron
Air consumption: 1.7 cubic ft/min. @ 80 psi
Airline Recommended: 1/4” I.D. tubing
Reservoir Capacity: 16 oz

Options

Troubleshooting the MD1200 Micro Drop Misting System

Most operating problems with the MD1200 Micro Drop Misting System can be traced to a simple cause such as air bubbles in the liquid lines or a plugged filter. The following guide will help troubleshoot the most common symptoms:

Symptom	Cause	Remedy
No or low liquid flow	Airflow setting too low, causing anti-siphon valve to activate	Increase airflow setting and check line pressure. Recommended at 60 psi/min.
	Liquid filter plugged	Replace filter.
Unit drips oil from the tip when OFF	Air in liquid lines	Re-prime unit.
Delay before lubricant flow starts	Air in liquid lines	Re-prime unit.
Air bubbles present in line	Unit not primed	Re-prime unit.
	Reservoir empty	Fill reservoir.
Weak airflow	Low supply pressure	Check line pressure. Recommended at 60 psi/min.
	Supply line too small	Supply line must be 1/4” I.D. min

Options

Coolubricator System

Coolubricator Systems are completely self-contained, positive-displacement, continuous spray systems that operate when the adjustable pulse generator automatically cycles the adjustable metering pump(s). Operators just have to supply an air signal to operate the system.

The liquid output can be adjusted with the brass knob located on each metering pump, and any amount of air can be used to atomize the liquid. A brass knurled air metering screw controls the flow of air atomizing out of the nozzle, which determines the density and distance of the spray. The Coolubricator System offers complete control of the spray as well as accurate placement of the lubricant or other material being applied. The spray output includes an approximately 15-20 degree angle depending on the amount of air introduced. Operators can adjust the area covered by using the air and liquid adjustments.

The Coolubricator System allows for multiple metering pumps to be stacked with each pump supplying its own nozzle. This configuration allows for greater spray coverage from multiple nozzles, which are ideal for multiple spindle drills and taps, stock punching, or any application requiring material being sprayed from several points. The nozzle outputs are controlled by independent adjustments of the air metering screw and metering pump stroke adjustment knob. Fluid output may be increased by using 3-drop output pumps in place of standard 1-drop pumps, though operators are encouraged to discuss this with the manufacturer beforehand.

Options

Operating the Coolubricator System

Operators of the Coolubricator System should be careful when installing and operating the auto misting system.

1. Determine the position and method of attachment – either directly or with optional magnets – before mounting the Coolubricator System to the application equipment.

2. Check for any obstructions that may pinch or kink liquid or air feed lines.

3. Mount the Coolubricator in close proximity to machine application either directly to the machine or with the optional magnets to the right and left flanges of the Coolubricator case or bottom of the case.

4. Mount and position the nozzles as desired.

5. Attach the air supply line to 1/4” FPT inlet on the air filter supplied with steel enclosure if enclosed and directly to the top manifold block if non-enclosed.

6. Make sure the reservoir or remote tank Low Level Switch is hardwired. This step is optional and can be skipped.

7. Wire the solenoid valve or connect the air pilot valve.

8. Fill the gravity feed reservoir or air trap.

9. Loosen the drain plug until liquid flows free of air.

10. Tighten the drain plug and cycle unit. On units supplied with fluid from a pressurized supply tank, set the supply tank pressure at 5-10 PSI and depress the air vent cap until the trap is 75% full.

11. Turn the brass adjustment knob to the maximum output setting and turn the air supply on and off with the optional manual ON/OFF valve or by manually depressing override cap. Operators may either start the machine tool with a remote switch, which was installed to energize optional electric solenoid, or activate the air signal to the air pilot valve.

12. Adjust the Pulse Generator to 6-8 pulses per minute.

13. Close the air metering screw when fluid is seen coming out of nozzle supply tube with each cycle. Open the air metering screw to the desired degree of atomization. Operators should use a minimum amount of air atomization to spread the lubricant and deliver it to the application area.

14. Use the Pulse Generator as the Primary Fluid Adjustment.

Options

Troubleshooting the Coolubricator System

Most operating problems with the Coolubricator System can be easily corrected. The operator should contact the manufacturer should any problem occur that is not addressed in this document.

Symptom	Cause	Remedy
No fluid output following reservoir evacuation	Pulse generator is set too slow or too fast.	1. Check the reservoir level. 2. Use full air line inlet pressure and check for stroking of the meter. Set to max. 3. Check for fluid movement in the capillary tubing at each meter stroke. 4. Loosen the 5/16” brass hex bleed screw 2 revolutions until only fluid comes out. Close the screw tightly and recheck the meter output. 5. Check inlet air and set to max.
Reduced fluid output	Non-lubricating fluids like water and solvents can cause premature wear of internal parts.	Check the air inlet pressure. Set to max so that meters operate crisply.
Pulse generator not functioning	Obstruction in the Coolubricator System	Remove the pulse generator and check the air inlet screen for particular matter.
Slow or sluggish meter operation	Broken spring on an inadequately lubricated U-cup	1. Make sure the air inlet pressure is set at max. 2. Rebuilt the meter.
Meter depressed and not returning	Piece of debris in the fluid causing the liquid pin to jam	1. Disassemble and reassemble the auto mister and look for broken parts. 2. Rebuild the meter if a return spring is broken.
Fluid flowing continuously out of the nozzle or Reservoir emptying without the system running	Check valve seal not properly functioning	Check the seat and seal spring for cleanliness.
Fluid accumulation in the outer tubing	Insufficient atomizing air exiting the nozzle	Increase the atomizing air flow or direct the nozzle coupler outlet downward to remove excess fluid.
Air bubbling upward into fluid reservoir	Possible leakage in the valve seat	1. Check the valve seat and seal for any signs of leakage. 2. Use a rebuild kit and replace the static sealing O-ring if necessary

Options

Innovation. Quality. Performance. Best in Industry. Worldwide.

Connections

top

Communication 2
Ethernet
- Ethernet Connection Information 2
- Setup for Ethernet Connection 8
- Ethernet Cables 10

Serial
- Setup for Serial Connection 11
- Communication Troubleshooting 14
- Communications Port Settings of Windows 16
- Serial Line Amplifier 18
- Serial Cable Construction 19
- Serial Communication 19

Communication

The ***MultiCam*** Controller will support both Ethernet and Serial communications; however, ***MultiCam*** recommends using Ethernet communications whenever possible. Connect

Ethernet Connection Information

TCP/IP Ethernet connection is the preferred form of communications to all ***MultiCam*** controllers. The following information discusses several key components of Ethernet connections and how those components relate to connecting personal computers (PCs) and ***MultiCam*** CNC machines.

Requirements

Any company choosing to utilize ***MultiCam*** Ethernet communications as recommended must meet the following system requirements:

- Network card installed in the computer
- TCP/IP software installed in the computer
- Ethernet 10/100 MHz multiple port switch (cannot be wireless or a router)
- Ethernet patch cabling
- ***MultiCam*** Controller IP/Subnet in the same class network (IP/Subnet) as the Host PC

Wireless communication is not recommended for any ***MultiCam*** Ethernet system.

Ethernet Process

Every computer on a network has an IP address. The IP address tells other computers that the Host PC is ready to communicate with another device on the network. Each computer IP address must be unique within that network (e.g., 192.168.0.176 is the IP address for only one computer on the network).

In addition to IP addresses, each computer on a network is also assigned a subnet address. The subnet address is simply a way of separating larger networks into smaller, more manageable ones (e.g., 255.255.255.0 is one subnet address for ***MultiCam*** operating systems). Additionally, ***MultiCam*** CNC machines by design can only be used on a class “C” Network.

Setup

When any technician visits a customer site, he or she must first determine what type of network is available for the connection. The 2 types of networks are described below.

- Dynamic Host Configuration Protocol (DHCP) Network – Automatic Configuration

  The DHCP server will automatically provide networked computers with the required TCP/IP configuration from a central location. When a computer requires the use of TCP/IP network resources, it broadcasts a request for address information. The DHCP server responds to this request by assigning a new address and sending it to the computer along with other required configuration information for the network. This information is acknowledged by the computer and used in setting up that configuration.

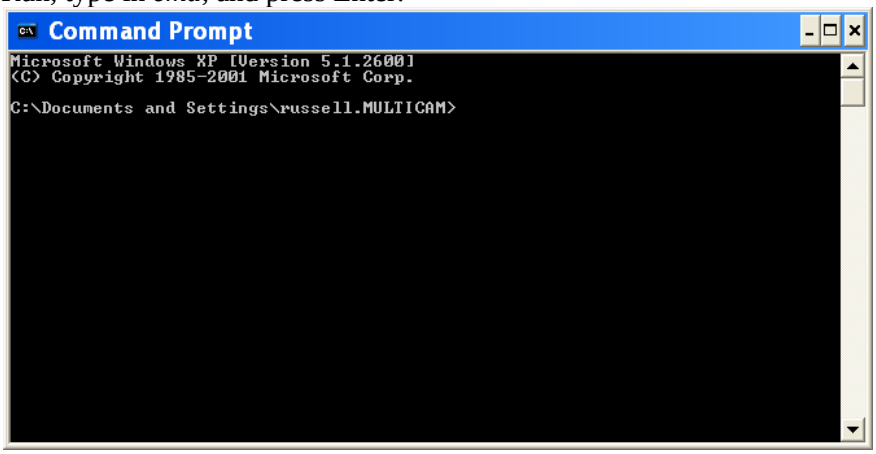
- Static Network – Manual Configuration

  The static network has a manually configured setup for each computer or device, and each device must be assigned the same subnet mask but a different IP address from all other computers and devices.

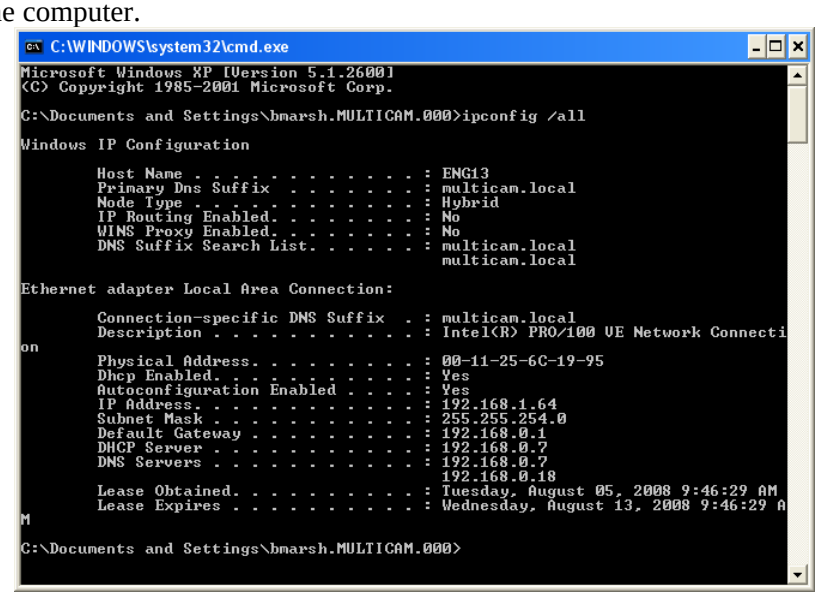
By default, most laptops and computers are shipped with DHCP already enabled. This does NOT mean that the network has a DHCP server that will automatically assign the computer an individual IP address. Technicians should still complete the following to determine the proper configuration for the machine:

1. Use the command prompt in Windows to check the IP configuration. Technicians can use either of the following methods:
   a. Go to Start >> All Programs >> Accessories >> Command Prompt.

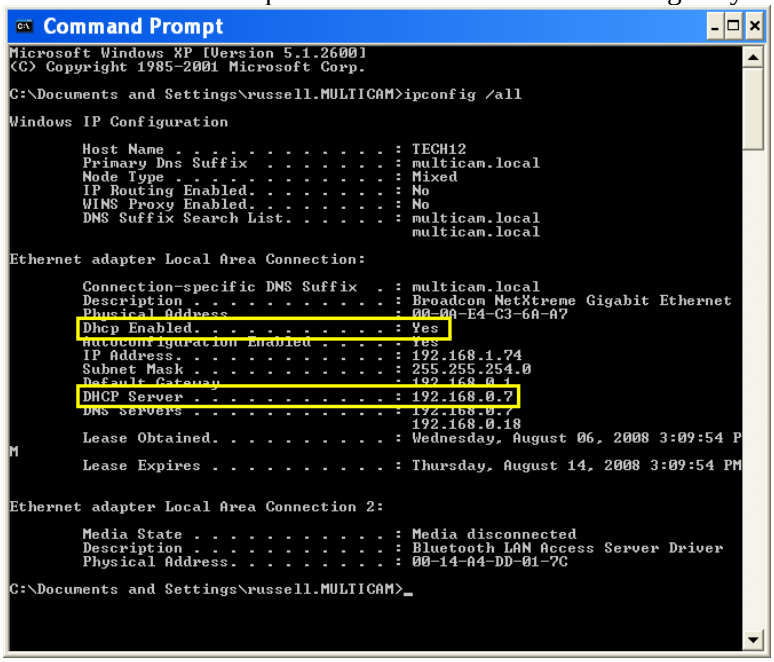
   b. Go to Start >> Run, type in *cmd*, and press Enter.



2. Type in *ipconfig /all* with a space between the *g* and the forward slash. This will pull up the Windows IP configuration for the computer.


C:\WINDOWS\system32\cmd.exe
Microsoft Windows XP [Version 5.1.2600]
(C) Copyright 1985-2001 Microsoft Corp.
C:\Documents and Settings\bmarsh.MULTICAM.000>ipconfig /all
Windows IP Configuration
Host Name . . . . . : ENG13
Primary Dns Suffix . . . . . : multicam.local
Node Type . . . . . : Hybrid
IP Routing Enabled. . . . . : No
WINS Proxy Enabled. . . . . : No
DNS Suffix Search List. . . . . : multicam.local
multicam.local
Ethernet adapter Local Area Connection:
Connection-specific DNS Suffix . : multicam.local
Description . . . . . : Intel(R) PRO/100 VE Network Connecti
on
Physical Address. . . . . : 00-11-25-6C-19-95
Dhcp Enabled. . . . . : Yes
Autoconfiguration Enabled . . . . : Yes
IP Address. . . . . : 192.168.1.64
Subnet Mask . . . . . : 255.255.254.0
Default Gateway . . . . . : 192.168.0.1
DHCP Server . . . . . : 192.168.0.7
DNS Servers . . . . . : 192.168.0.7
192.168.0.18
Lease Obtained. . . . . : Tuesday, August 05, 2008 9:46:29 AM
Lease Expires . . . . . : Wednesday, August 13, 2008 9:46:29 A
M
C:\Documents and Settings\bmarsh.MULTICAM.000>

3. Look for the IP address and subnet mask in the *ipconfig /all* screen (i.e., DHCP Enabled status and the DHCP Server address). This information will be required if the technician has to change any of the addresses.


Command Prompt
Microsoft Windows XP [Version 5.1.2600]
(C) Copyright 1985-2001 Microsoft Corp.
C:\Documents and Settings\russell.MULTICAM>ipconfig /all
Windows IP Configuration
Host Name . . . . . : TECH12
Primary Dns Suffix . . . . . : multicam.local
Node Type . . . . . : Mixed
IP Routing Enabled. . . . . : No
WINS Proxy Enabled. . . . . : No
DNS Suffix Search List. . . . . : multicam.local
multicam.local
Ethernet adapter Local Area Connection:
Connection-specific DNS Suffix . : multicam.local
Description . . . . . : Broadcom NetXtreme Gigabit Ethernet
Physical Address. . . . . : 00-0A-E4-C3-6A-A7
Dhcp Enabled. . . . . : Yes
Autoconfiguration Enabled . . . . : Yes
IP Address. . . . . : 192.168.1.74
Subnet Mask . . . . . : 255.255.254.0
Default Gateway . . . . . : 192.168.0.1
DHCP Server . . . . . : 192.168.0.7
DNS Servers . . . . . : 192.168.0.7
192.168.0.18
Lease Obtained. . . . . : Wednesday, August 06, 2008 3:09:54 P
M
Lease Expires . . . . . : Thursday, August 14, 2008 3:09:54 PM
Ethernet adapter Local Area Connection 2:
Media State . . . . . : Media disconnected
Description . . . . . : Bluetooth LAN Access Server Driver
Physical Address. . . . . : 00-14-A4-DD-01-7C
C:\Documents and Settings\russell.MULTICAM>

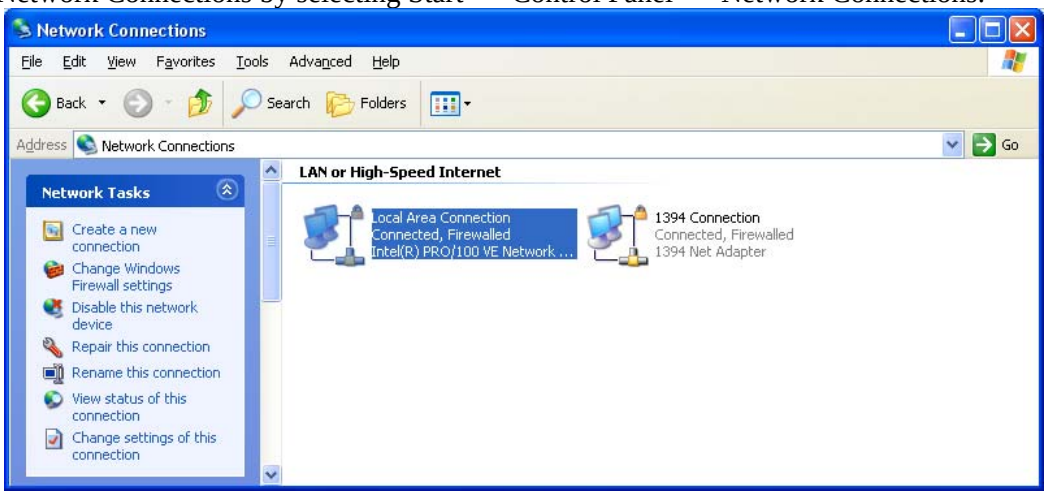
- If the network has an active DHCP, then DHCP Enabled will show *Yes*. The technician must also have an assigned IP address for the DHCP server (shown above).
- If the network does not have an active DHCP, then DHCP Enabled will show *No*, which indicates a static network.
- If any of the above information is not provided, then the technician is working with a static network.

Static Network Changes

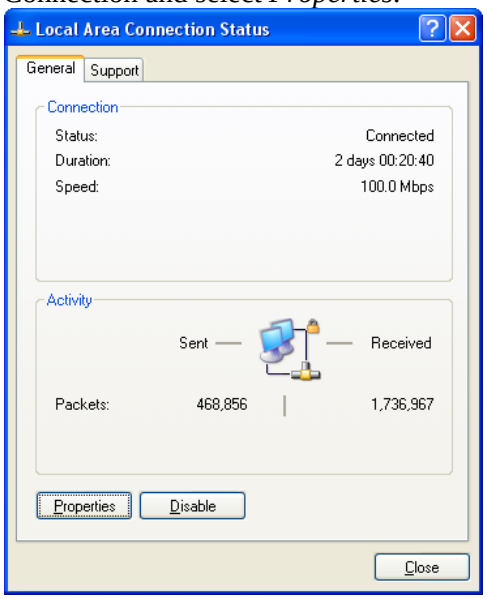
Technicians working with a static network may have to change the static IP and/or subnet address of the computer, the controller, or both. The following directions describe how technicians can change the IP and subnet addresses:

Computer Static IP/Subnet Address

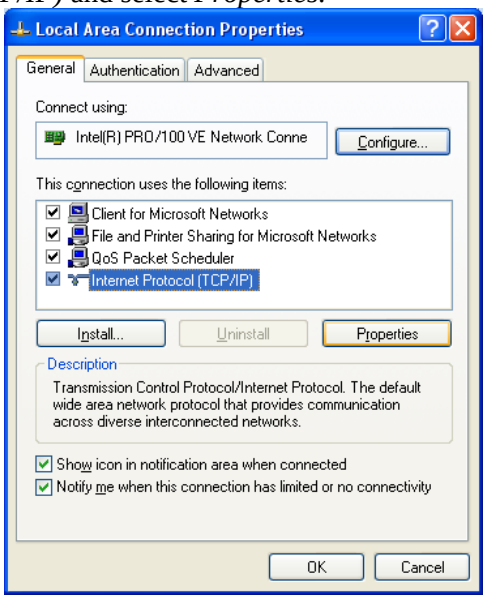
1. Access Network Connections by selecting Start >> Control Panel >> Network Connections.



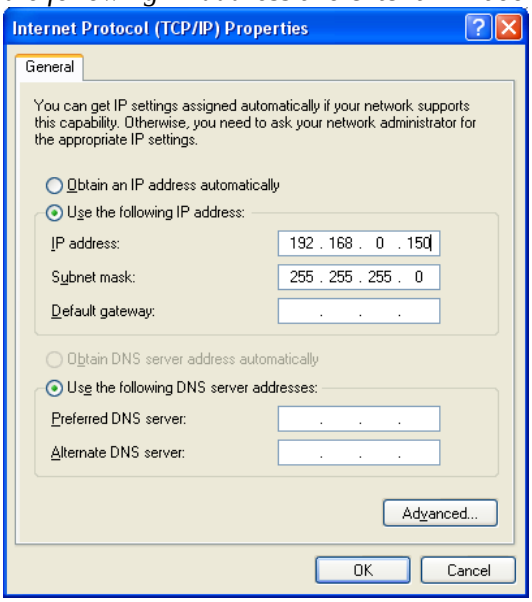
2. Double-click on the Local Area Connection and select *Properties*.



3. Highlight *Internet Protocol (TCP/IP)* and select *Properties*.



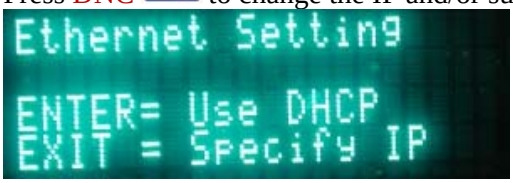
4. Click in the circle next to *Use the following IP address* and enter an IP address and the subnet mask.


Internet Protocol (TCP/IP) Properties
General
You can get IP settings assigned automatically if your network supports this capability. Otherwise, you need to ask your network administrator for the appropriate IP settings.
Obtain an IP address automatically
Use the following IP address:
IP address:
192 . 168 . 0 . 150
Subnet mask:
255 . 255 . 255 . 0
Default gateway:
Obtain DNS server address automatically
Use the following DNS server addresses:
Preferred DNS server:
Alternate DNS server:
Advanced...
OK
Cancel

Controller Static IP/Subnet Address

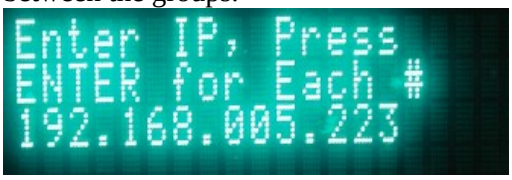
1. Press Shift, Help, and Two to access the Ethernet Information screen on the keypad.

2. Press DNC to change the IP and/or subnet mask address.



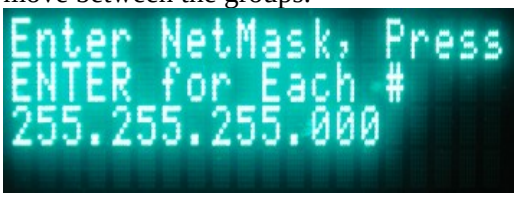
3. Press Exit to specify the IP address.

4. Type in the IP address as 4 groups of 3 numbers (e.g., 192.168.005.223) and press Enter to move between the groups.



5. Press Enter and wait for the screen to prompt for the netmask.

6. Type in the netmask address as 4 groups of 3 numbers (e.g., 255.255.255.000) and press Enter to move between the groups.



7. Press Enter to save the changes.

8. Reboot the machine for the IP and/or subnet mask address changes to take effect.

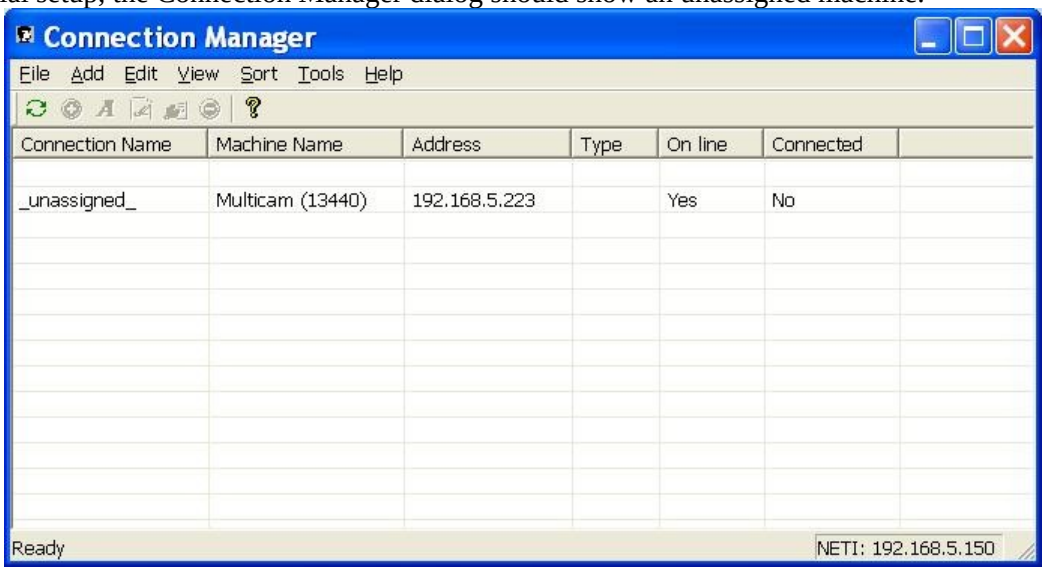
Connect

Setup for Ethernet Connection

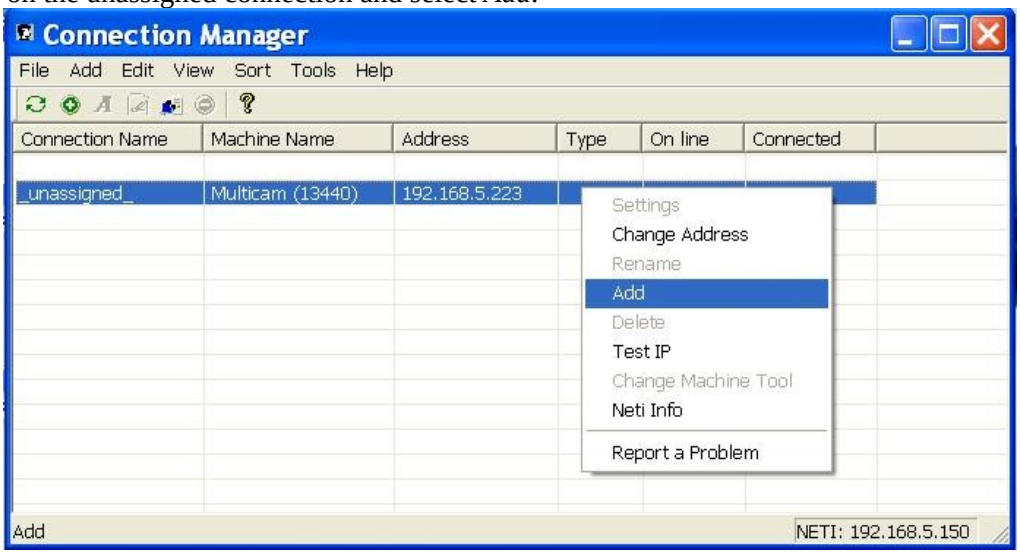
The Ethernet communication is set up through the Machine Tool Properties applet. Operators should make sure that the CNC machine is on and connected to the computer via an Ethernet connection before attempting to automatically install the Ethernet connection. The following directions assume that the operator has complied with all hardware requirements for Ethernet connections (see Ethernet Connection Information).

1. Right-click on the Machine Connections icon in the toolbar and select Connection Manager.

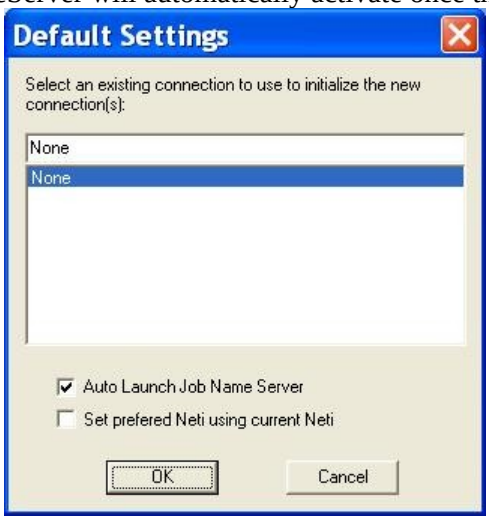
At the initial setup, the Connection Manager dialog should show an unassigned machine.



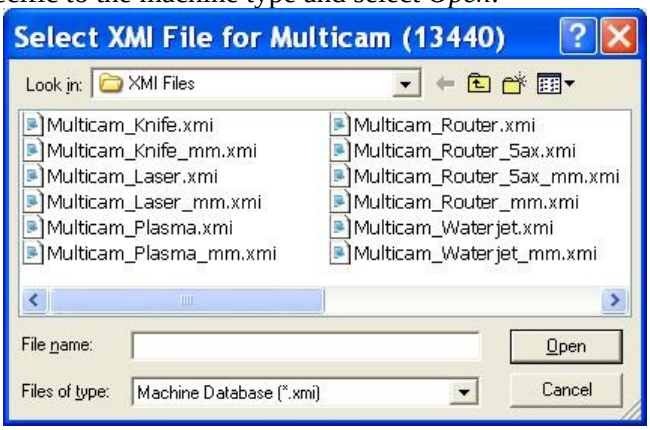
2. Right-click on the unassigned connection and select *Add*.



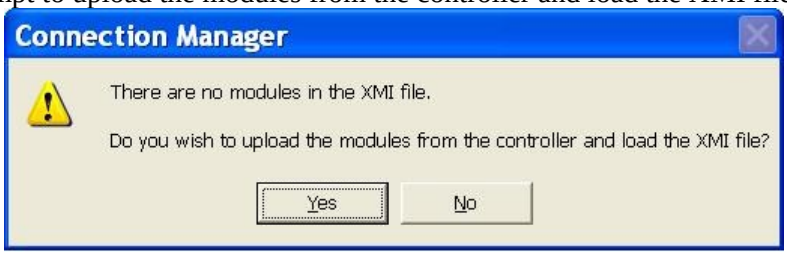
3. Select *None* at the Default Settings dialog. All customers are encouraged to check the box next to *Auto Launch Job Name Server* so that JobNameServer will automatically activate once the machine is connected.



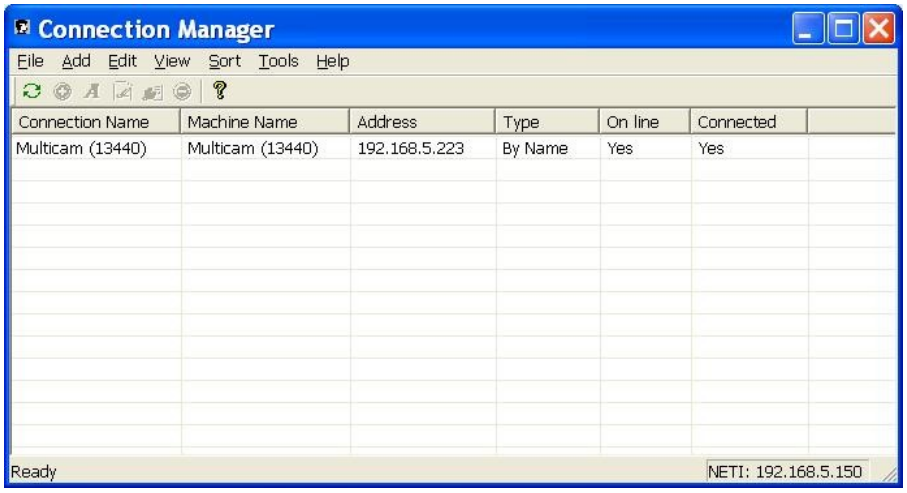
4. Highlight the XMI files specific to the machine type and select *Open*.



5. Select *Yes* at the prompt to upload the modules from the controller and load the XMI file.


Connection Manager
There are no modules in the XMI file.
Do you wish to upload the modules from the controller and load the XMI file?
Yes
No

This will assign a connection name to the machine.



Connect

Ethernet Cables

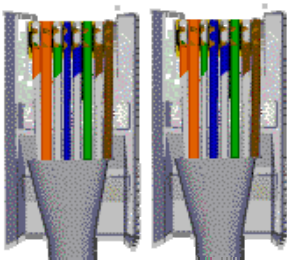
If the company has an established network, then the computers are probably already connected through a hub or switch. A patch cable is a standard cable used to connect a network device consisting of a computer, a network printer, and a ***MultiCam*** Controller to a hub or switch. However, the patch cable is unable to provide a direct connection with the ***MultiCam*** Controller, so the host PC must be connected through a switch or hub.

RJ45 Patch Cable

A patch cable is also known as a straight-through cable. The wires are in the same order on connector 1 as they are on connector 2. The RJ45 connectors are used on each end of the cable, and operators must verify that the connectors are being held the same way when checking.

Connect

Connector 1		Connector 2
Pin 1	>>>	Pin 1
Pin 2	>>>	Pin 2
Pin 3	>>>	Pin 3
etc...	>>>	etc...

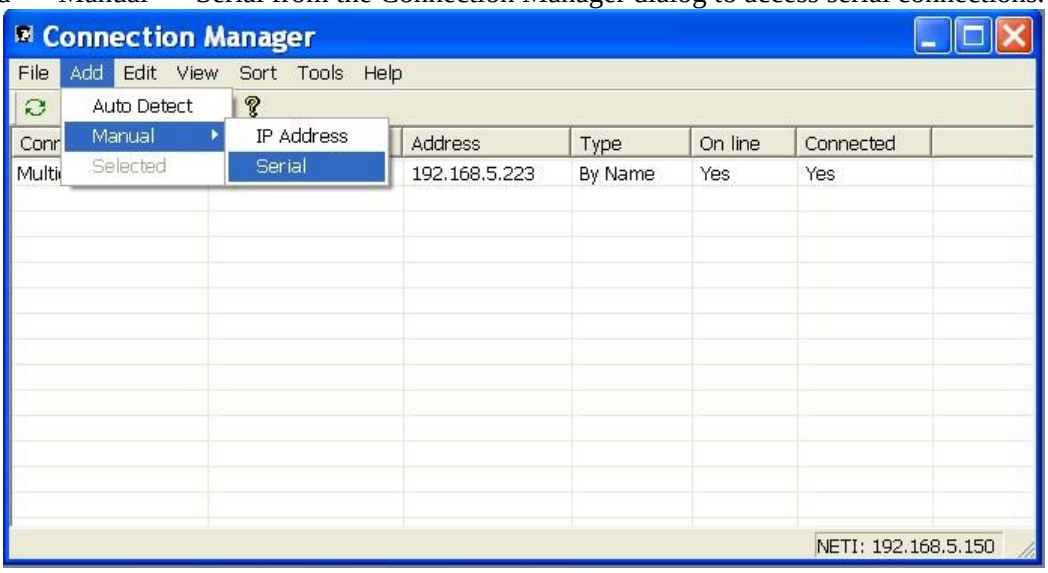


Setup for Serial Connection

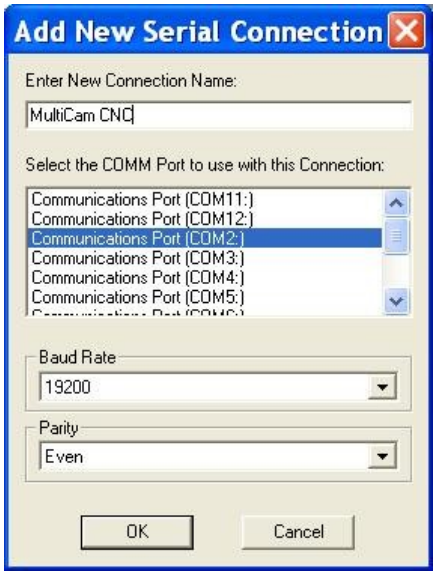
The serial communication is set up through Machine Tools. Ethernet communication is always recommended over serial, but operators may need to use serial connections for troubleshooting or quick-fixes for communication.

1. Right-click on the Machine Connections icon in the toolbar and select Connection Manager.

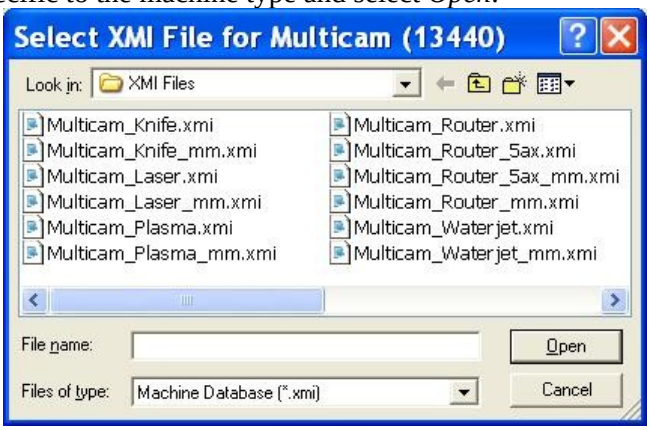
2. Select Add >> Manual >> Serial from the Connection Manager dialog to access serial connections.



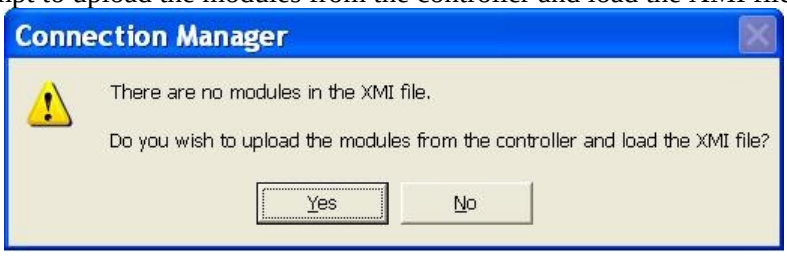
3. Enter or select the following information and select *OK*:
   - Connection Name (e.g., MultiCam CNC)
   - Communications port (e.g., COM2)
   - Baud Rate – 19200
   - Parity – Even



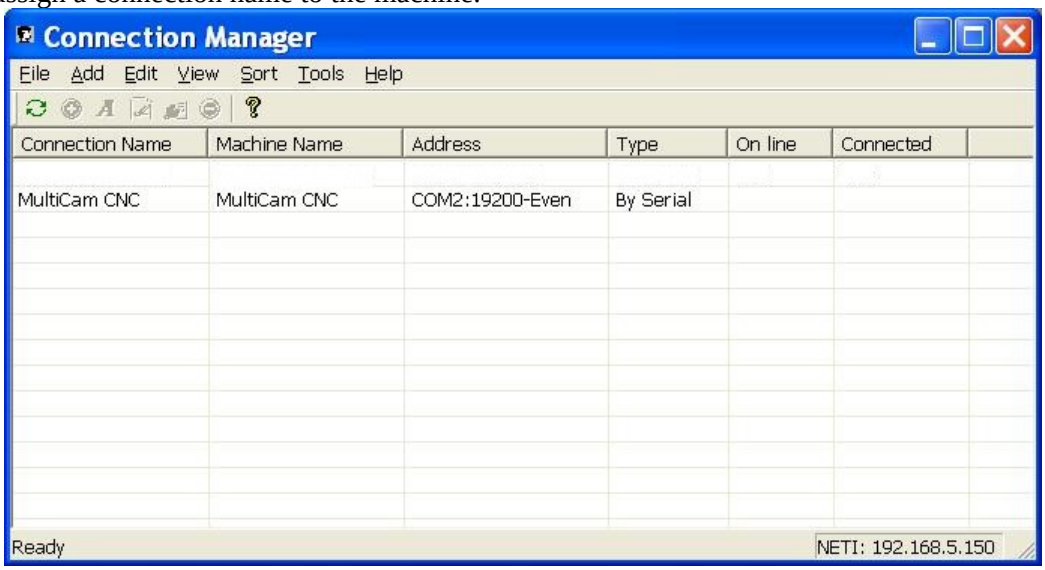
4. Highlight the XMI files specific to the machine type and select *Open*.



5. Select *Yes* at the prompt to upload the modules from the controller and load the XMI file.


Connection Manager
There are no modules in the XMI file.
Do you wish to upload the modules from the controller and load the XMI file?
Yes
No

This will assign a connection name to the machine.



Connect

Communication Troubleshooting

The ***MultiCam*** CNC machine is shipped with no connectors, and operators should purchase a 25' serial cable containing 9 wires to connect the ***MultiCam*** Controller to the Host PC. Data is sent from the computer to the machine controller. The data leaves the computer, passes through the computer Com Port or serial port, and moves through the serial cable and into the controller. Occasionally, operators experience difficulty when the machine does not accept a file, is unable to access DNC, loses connection, never establishes connection, or cuts off in a straight line in a manner that is not part of the original output file.

Most Common Reasons for Communication Failure

If the machine works some of the time but experiences random problems, then the machine may be experiencing some sort of communication problem. Here is a list of the most common problems for communication failure. Operators are encouraged to refer to this file before calling the local support technician or the ***MultiCam*** Technical Support.

* The Windows Com Port settings, especially the FIFO buffer, are not correct. See Communications Port Settings of Windows.

* The serial cable is longer than 100ft and is used without a Line Amplifier. See Serial Line Amplifier.

* The serial cable has been operated next to high voltage or fluorescent lights.

* The cable has been nicked, cut, or damaged in some fashion. See Serial Cable Construction.

* The wrong Com Port has been selected on the host PC.

* The Com Port IRQ conflicts with a modem installed in the host PC.

* The internal ribbon cable inside the machine control cabinet has been damaged or has lost its connection.

* The Max202 chip on the control board has failed.

Troubleshooting

The process of elimination is often used when troubleshooting various communication problems as more than one problem may have caused the current machine difficulty and different types of machines will respond differently to similar communication problems. The serial ports in the machine and in the computer could be damaged by a bad electrical storm or a comparable natural occurrence, and this damage will not always be fixed the same way for each machine. Below are common sites for communication problems and suggested ways to repair the damage.

Port Settings should be checked to verify that port settings and communication settings are correct.

* Machines that were manufactured after January 1996 have the following communication settings:
  Baud rate: 19200; parity: even; data bits: 8; stop bits: 1

* Machines that were manufactured before January 1996 have the following communication settings:
  Baud rate: 9600; parity: none; data bits: 8; stop bits: 1

Cable Testing 1 can be started with unplugging the serial cable from the side of the machine and plugging it back in before unplugging the serial cable from the back of the computer and plugging it back in. If the communications are still not working, Cable Testing 2 may help.

Cable Testing 2 involves opening the control box to inspect the cable connection extending from the side of the box where the external serial cable is plugged into the control board. This cable will be a ribbon cable plugged into the control board labeled G960, M10, M23 or M24. The connection going into the board can be unplugged and plugged back in. On many systems, there is an additional small ribbon cable extending from the top control board to the bottom control board that can also be tested. If the communications are still not working, Computer Port Testing 1 may help.

Computer Port Testing 1 for a computer with 2 serial ports involves plugging the machine into the other serial port and changing the port setting in DNC or the machine file output software. **Computer Port Testing 1** for a system with a serial port mouse involves plugging the mouse into the old port associated with the CNC machine. Windows should find the mouse automatically. If the machine works and the mouse does not, then the computer Com Port is probably bad and will need to be repaired by the computer vendor. If the communications are still not working, Computer Port Testing 2 may help.

Computer Port Testing 2 involves unplugging the serial cable from the side of the machine control box and plugging the loop-back plug provided in the tool box into the open end of the serial cable. After going to a DOS prompt, the operator should change directories to the mc directory (cd\mc) and run COMCHK.EXE. This will test the serial port and the cable. If the communication status is bad, errors will display on the screen. These errors could point to a faulty cable or a computer serial port. The test can be ended by pressing the ESC key. If the communication status is good, a continuous line of dots will appear along the bottom of the screen. If this happens, Machine Port Testing may help.

Machine Port Testing involves unplugging the serial cable from the computer and plugging the loop-back plug provided in the tool box into the open end of the serial cable. The collet and cover nut from the spindle should be removed, and the machine should be turned off and back on. If the machine enters self-test and begins to move, then the machine communications and serial cable are okay. At this point, the operator can probably assume there must be a problem with the computer Com Port or associated settings. If this process does not work, then 1 of 3 things have happened:

1. The serial port chip is bad and needs to be replaced. ***MultiCam*** can be contacted for the chip and installation instructions.

2. The internal ribbon cable connections have been damaged.

3. There is no self-test file loaded in the controller. ***MultiCam*** can be contacted for additional Help.

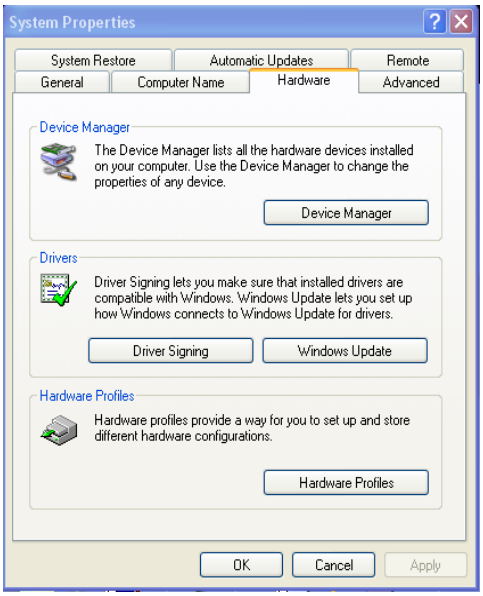
Connect

Communications Port Settings of Windows

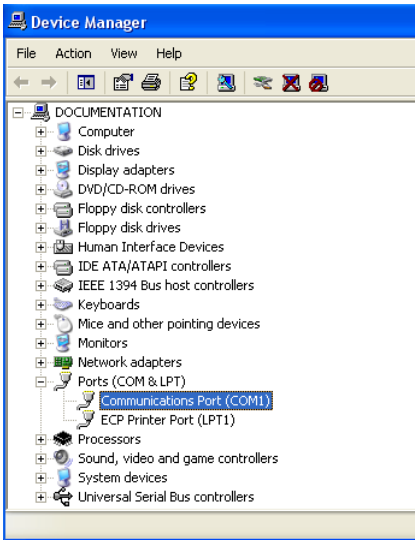
Most PCs are running Windows 2000 or higher as the Operating System. The default configuration of Windows is set to take advantage of larger (16 byte) FIFOs in the COM ports.

The ***MultiCam*** Controller has 4 to 7 character-receive FIFOs, which is less than half the standard default setting. Operators should reduce their Transmit FIFO levels when using a ***MultiCam*** Controller to lessen the chance for poor communications or loss of data that would result from having the settings too high.

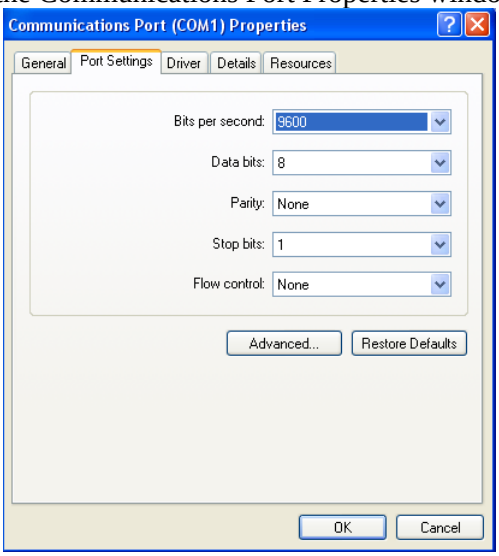
1. Right-click on the My Computer icon from the Windows Desktop and select the Hardware tab. Select *Device Manager*.


System Properties
System Restore
Automatic Updates
Remote
General
Computer Name
Hardware
Advanced
Device Manager
The Device Manager lists all the hardware devices installed on your computer. Use the Device Manager to change the properties of any device.
Device Manager
Drivers
Driver Signing lets you make sure that installed drivers are compatible with Windows. Windows Update lets you set up how Windows connects to Windows Update for drivers.
Driver Signing
Windows Update
Hardware Profiles
Hardware profiles provide a way for you to set up and store different hardware configurations.
Hardware Profiles
OK
Cancel
Apply

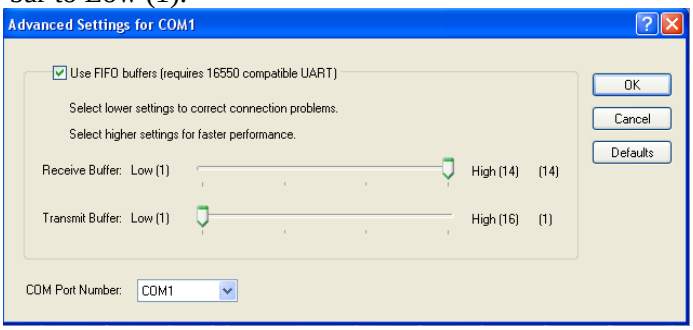
2. Select Ports (COM & LPT) and then select Communications Port (COM1) or the port to which the ***MultiCam*** Controller serial cable is connected.



3. Select the Port Settings tab from the Communications Port Properties window and select *Advanced*.



4. Drag the Transmit Buffer bar to Low (1).


Advanced Settings for COM1
Use FIFO buffers (requires 16550 compatible UART)
Select lower settings to correct connection problems.
Select higher settings for faster performance.
Receive Buffer: Low (1) High (14) (14)
Transmit Buffer: Low (1) High (16) (1)
COM Port Number: COM1
OK
Cancel
Defaults

5. Select *OK* to all the windows that were opened and restart Windows to activate this setting. Connect

Serial Line Amplifier

MultiCam recommends using the Bravo Line Amplifier or a comparable serial line amplifier and requires a line amplifier when the serial cable reaches or exceeds 100'. Not using the line amplifier will create communication problems between the host PC and the ***MultiCam*** CNC machine as too much load will be placed on the Com Port as well as on the Max202 chip of the ***MultiCam*** Controller.

The installation of the serial line amplifier involves the AC power adapter, line amplifier, DB25 female to DB9 male adapter, a 6' adapter cable with a DB25 male connector and DB25 female connector, and a serial cable with a DB9 male connector and DB9 female connector.

1. Flip the line amplifier over and make sure that the 2 switches are positioned towards the middle of the line amplifier.

2. Place the DB25 male connector of the 6' adapter cable into the "To DTE" end of the line amplifier.

3. Place the DB25 female connector on the other end of the 6' adapter cable into an available Com Port of the PC.

4. Place the DB25 female end of the DB25 female to DB9 male adapter into the "To DCE" end of the line amplifier.

5. Place the DB9 female end of the serial cable into the DB9 male end of the DB25 female to DB9 male adapter.

6. Insert the AC power adapter plug into the socket on the line amplifier and plug it into an AC outlet.

Connect

Serial Cable Construction

Operators should avoid installing serial cables next to high voltage lines and should prevent any foot traffic from occurring over or across the serial cables. The serial cable should not be involved in a situation where damage is probable, and operators who create their own serial cables should not exceed 100'. Serial cables that exceed 100' require the assistance of line amplifiers, and ***MultiCam*** has tested lengths of serial cable up to 1000' that utilized serial line amplification.

The following is the pin out for the ***MultiCam*** serial cable. In both cases, Pin #9 is not used.

Computer End, DB9 Female

Pin #	*Color*
1	(not used)
2	Red
3	Blue
4	White
5	Yellow
6	Black
7	Orange
8	Brown

Machine End, DB9 Female

Pin #	*Color*
1	(not used)
2	Blue
3	Red
4	Black
5	Yellow
6	White
7	Brown
8	Orange

Connect

Serial Communication

The ***MultiCam*** Controller can communicate with a host PC through an RS232 Serial or Com Port. The serial port transmits one bit at a time spread out serially over time. Since only 1 bit rather than 8 is transmitted at a time, only 1 data line exists in the cable for each direction of travel with 2 lines total. A half-duplex allows only 1 data line enabling communication in 1 direction at a time, while the full-duplex allows 2 data lines enabling simultaneous communications in both directions. Thus, a serial port cable is thinner than a parallel port cable.

The port is called an RS232 port because the electrical characteristics follow the EIA Recommended Standard #232, which specifies the operating voltage (± 12V), relative to a common reference. Other details, such as impedance and driver capacity, are specified in this standard.

The controller's default configuration for options within the standard is 8 data bits, 1 stop bit, no parity, full-duplex, and hardware handshaking. Baud rates supported are 9.6K, 19.2K, 38.4K, and 57.6K bits per second with the standard setting being 19.2K.

The controller operates better when the device transmitting the motion commands obeys DTR hardware handshaking. The sophisticated processing and motion firmware provides continuous motion under most circumstances. In the event that additional processing time is required by the controller and the incoming character buffer fills, the DTR handshaking line will go low to prevent further transmissions of commands from the host PC. When the controller has processed enough of the incoming character buffer, it will then raise DTR and the host PC will resume transmitting.

Connect

Innovation. Quality. Performance. Best in Industry. Worldwide.

Software

top

Software 2
- AlphaCam Post Processor Installation 2
- EnRoute 3 Driver Setup 4
- EnRoute 3 View and Preference Settings 8

MultiCam Software 13
Motion Controller Utilities 13
PSS Motion Controller Utilities
- JobServer 14
- JobPreviewer 15
- JobReporter 16
- MotionMechanic 17
- Machine Connections 17

Suite4 Motion Controller Utilities 18
- JobNameServer 18
- JobEditor 19
- JobReporter 19
- MotionMechanic 20
- ConnectionManager 20
- JobConsole 21

JobServer Programs
- Digitize 22
- HotWatch 22
- JobQueue 22
- Control Settings 23
- Project Settings 24
- File Settings 25
- CNC Configuration 26
- HPGL Configuration 27
  - HPGL Command Support 28
  - HPGL Commands 30
- Job Estimating 31

MultiCam M- and G-Code Support 32
Mapping M Code to Output 34
Glossary 36

Software

The ***MultiCam*** Router can use files created from many popular software packages in the industry, such as AlphaCam and EnRoute, as well as files created in G Code and HPGL. The productivity suite enhances communication between the ***MultiCam*** Router and the host computer. Software

AlphaCam Post Processor Installation

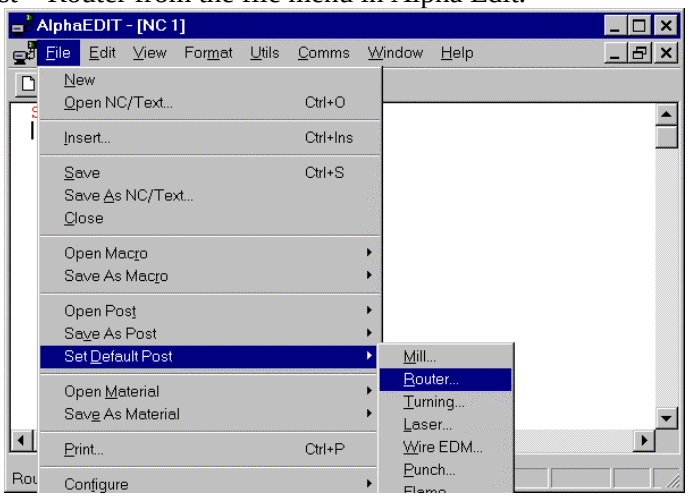
MultiCam offers 2 AlphaCam post processor systems for use depending on the type of post system in the machine. The ***MultiCam*** ATC-ARP post processor interfaces with both ATC and multiple head systems while the ***MultiCam*** ARP post processor interfaces with single head systems and outputs a manual tool change command so that multiple tool change files can be run on single head systems.

Multiple Post Systems

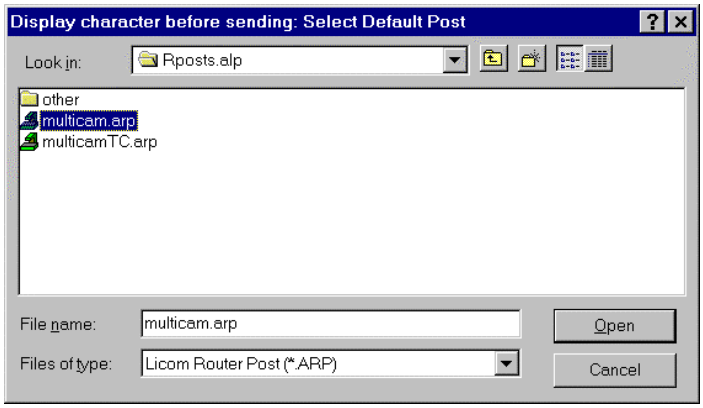
1. Copy the appropriate post processor located in Software Drivers\AlphaCam from the ***MultiCam*** Installation CD into the LICOMDAT\RPOSTS.ALP directory using Windows Explorer.

2. Start the Alpha Edit program from the AlphaCam folder in the Windows Start menu once the post processor is copied.

3. Select Set Default Post – Router from the file menu in Alpha Edit.



4. Select the desired post processor from the list. Click *Open*. This will set the selected post processor as the default.



Single Post Systems

1. Contact ***MultiCam*** or one of its authorized distributors for a copy of the OnePost file to use with the single post system.

2. Copy the encrypted OnePost file into the LICOMDAT\RPOSTS.ALP directory. This file has no extension.

3. Verify that AlphaCam completes the auto detect of the single post system post processor once the file is copied into the RPOSTS.ALP directory.

Innovation. Quality. Performance. Best in Industry. Worldwide.

EnRoute 3 Driver Setup

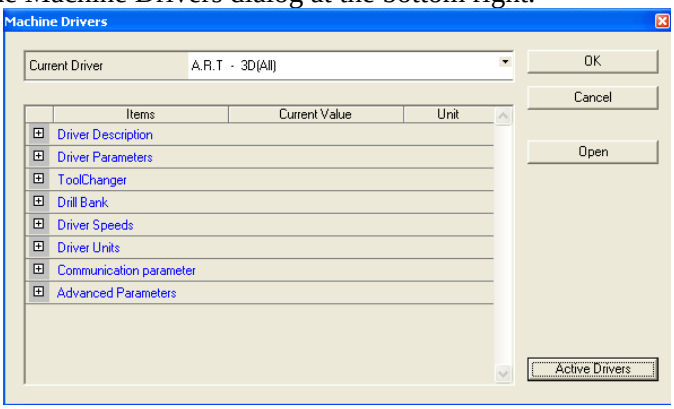
The EnRoute software package is used to assist operators in creating job files for the ***MultiCam*** Router. Each operator must have both an installation CD to set up the capability and a dongle to connect with the host PC each time the program is run.

Verifying Computer Compatibility

1. Refer to the EnRoute Recommended System Requirements to make sure the host PC is compatible with the software package.

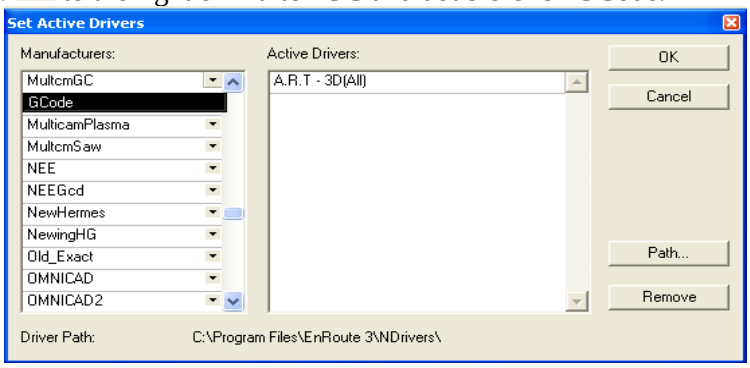
2. Open EnRoute and select Setup/Machine Setup from the toolbar at the top.

3. Click *Active Drivers* in the Machine Drivers dialog at the bottom right.



4. Scroll down on the left side to MultcmGC in the Manufacturers Window.

5. Click on the down arrow to the right of MultcmGC and double-click GCode.



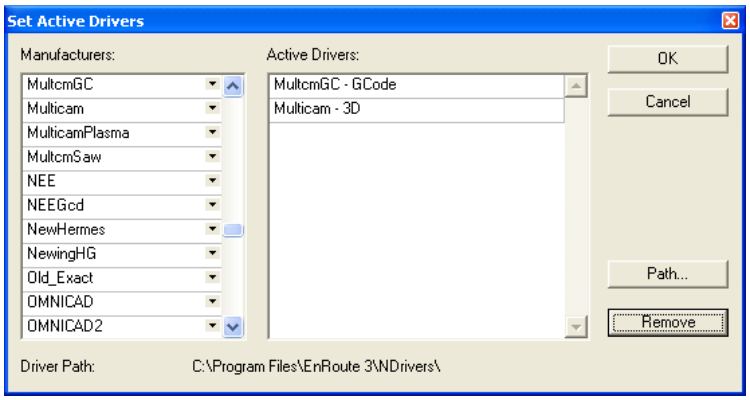
This will add MultcmGC/GCode to the active drivers list on the right.

Selecting the Driver

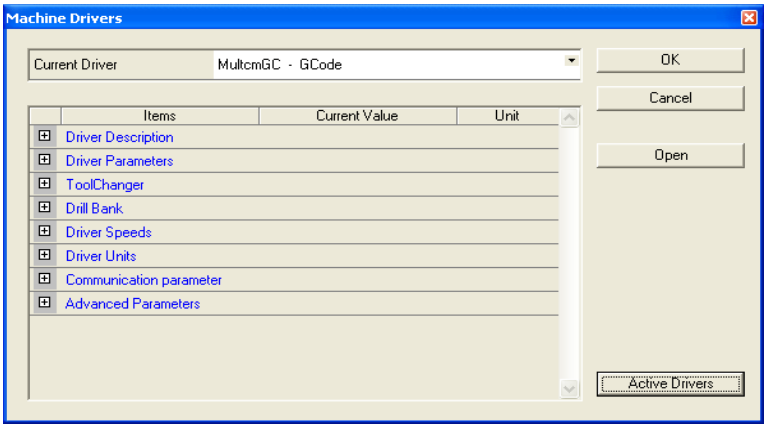
1. Select the appropriate driver based on the system requirements (e.g., Multicam - 3D Driver [HPGL]).
   a. **Multicam – 2D** uses the ***MultiCam*** 2D Driver for machines that **do not** support 3D and will override all cut parameters set at the keypad.
   b. **MultiCam – 3D** uses the ***MultiCam*** 3D Driver for machines that **do** support 3D and can be used for cutting bridges, multiple-pass cuts, or 3D machining. This driver will output all cut parameters (e.g., X,Y,Z cut speeds, depth, tool lift, spindle speed, and dwell) and will override any parameters set at the keypad.
   c. **MultiCam – No Depth** can be used on 2D and 3D ***MultiCam*** machines and will only output the tool path without outputting any cut parameters (e.g., X,Y,Z cut speeds, depth, tool lift, spindle speed, and dwell) regardless of their settings in EnRoute. All settings must be set on the keypad pendant. This driver will not output 3D.

2. Highlight the default driver (e.g., A.R.T. 3D [All]) and select *Remove*.

3. Select *OK* to accept the appropriate ***MultiCam*** GC/GCode driver.



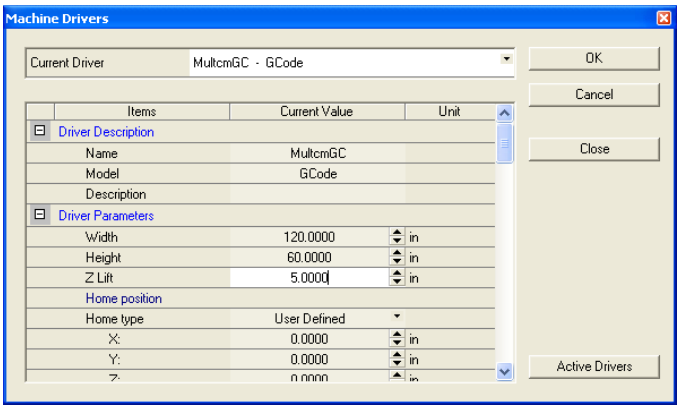
4. Select *OK* to return to the Machine Drivers dialog. The selected driver should be listed at the top.



Adjusting Table Settings

1. Select *Open* to adjust the table settings.

2. Scroll down to Driver Parameters and change the width, height, and Z lift to match the table.
   a. Width = X-axis (front to back; long side of the table)
   b. Height = Y-axis (left to right; short side of the table along the gantry)
   c. Z lift = Z-axis (up and down; Tool Lift Height above the table)

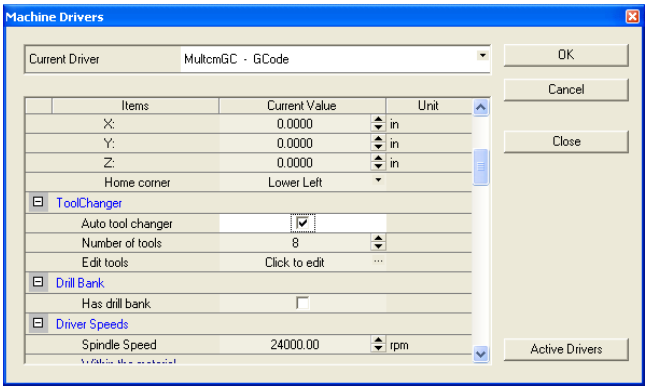


3. Make sure the dimensions are correct before moving on.
   a. If there is no ATC attached to the Router, the operator may select *OK* and skip ahead to the **Completing EnRoute Setup** section.
   b. If there is an ATC attached to the Router, the operator should configure the tool changer.

Configuring the Tool Changer

1. Scroll down to Tool Changer and check the box next to Auto Tool Changer.

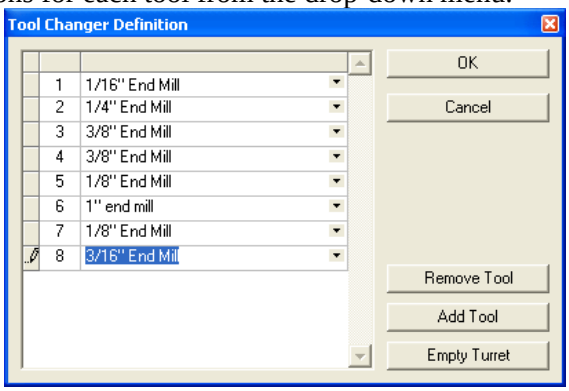
2. Use the double arrows next to Number of Tools to correct the file to show 8 for a carousel or 4 for a linear tool changer. The numbers 4 and 8 do **not** have to match the total tools in the carousel.



3. Double-click the *Click to Edit* section located below the selected number of tools and next to Edit tools.

4. Determine how the tools will be set up in the Tool Changer and click on the first down arrow .

5. Select the correct dimensions for each tool from the drop-down menu.



6. Select *OK* when finished.

Completing EnRoute Setup

1. Select *OK* in the Machine Driver Dialog to finish the EnRoute Setup.

2. Make no changes to the Driver Units section as the system is configured with default settings and will not work if these settings are changed. The operator can modify the settings under *Preferences* or the view setup F10.

Software

EnRoute 3 View and Preference Settings

While ***MultiCam*** recommends different settings for standard configurations, the operator is able to review the condition of the cutting sequence and the setup before the cutting sequence begins.

View Options

* Check the screen views to view the associated rulers. The rulers are used to verify size and positioning relative to the location in that Layout View.

* Scroll up, down, left, or right to pan the viewing area without moving the contours.

Toolbars

* Select Setup/Toolbars to choose a toolbar in order to view or hide it. All toolbars will be shown by default.

* Move a toolbar to the desired location by holding down the left mouse button on the dividing line that separates each bar and dragging it. Once the toolbar is close enough to one of the outer edges of the EnRoute window, the toolbar will snap to it, rotating itself accordingly, and the operator may simply release the mouse button.

* Drag the toolbar to the layout viewing area and release it to the title bar at the top. The toolbar can then be grabbed by clicking and holding down the left mouse button.

* Return to the default settings by selecting Setup/Toolbars/Reset to Defaults.

The Preference Settings available in EnRoute 3 (i.e., General, Initialization, Display, Units, Grids, View Setup, Start Points) are used to maximize or speed up the process in which the operator creates a design and applies the tool paths. While the operator has the final say in setting up the program, ***MultiCam*** recommends the following configurations. More information on a particular setting is available by pressing F1 Help, selecting the Search tab, typing in the tab name (e.g., General tab), and double-clicking the tab name under List Topics.

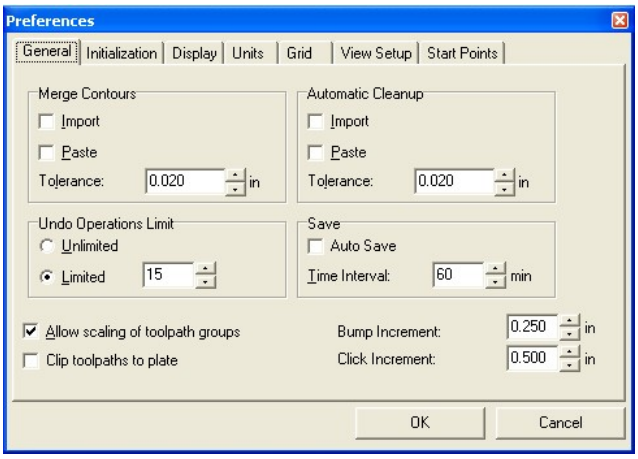
General

* Uncheck Merge Contours and Automatic Cleanup options since they may distort or join contours. Both the Merge and Cleanup options, found under Transform, can be completed once the artwork is successfully brought in.

* Undo Operations Limit due to PC resources and slow operation time. The Limited section should be set around 15 or 20 undos, though the operator would probably be better off starting over if he or she must undo more than 10 times.

* Undo Allow Scaling of Toolpath Groups to prevent using up resources. Toolpath is included in the size when scaling these groups, and it may be better to delete the tool paths and rescale the contours.

* Determine where Clip toolpaths to plate is necessary since it allows the operator to output only the tool path groups that are on the plate. This is useful when creating jobs that are too big for one sheet of material but should be kept together. An out-of-bounds message can occur when the operator is running a file as a closed contour, though parts of the file are not cutting, because either the tool path option is checked or the tool path is hanging off the plate.



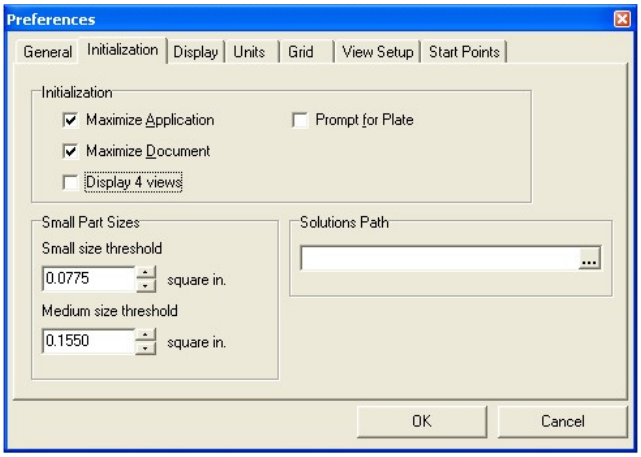
Initialization

* Select Maximize Application and Document to open the program and any files at maximum window size.

* Reserve the Display 4 Views option until toolpaths are in place and a particular view should be seen normally. Display 4 Views will show the top, front, side, and perspective view when opening, importing, or starting a new file.

* Select Prompt for Plate so that the system will prompt at the start of a file to define a plate or material size. A plate does not have to be defined and can be handled when nesting objects or running the 3D simulation. The Clip to Plate option should be unchecked when not defining a plate.

* Avoid changing the default settings for Small Part Sizes.

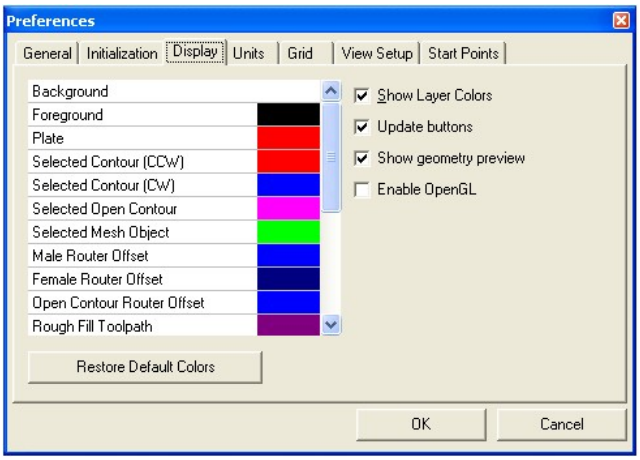


Display

* Avoid changing the Display Settings from the default setting.

* Double-click the individual Colors to make any preferential changes for contour appearance during viewing.

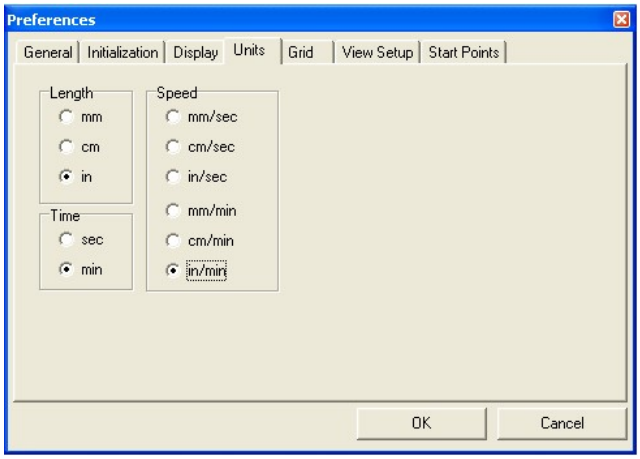
* Leave the 3 options to the far right checked in order to make certain commands user friendly and more functional.



Units

* Adjust the Length Time and Speed only when working with metric systems as most companies in the US use inches per minute, which is also the basis for most tool catalog formulas.

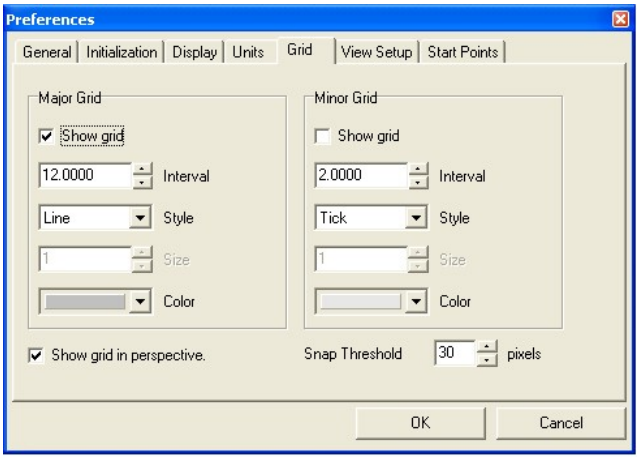
* Change unit measurements in this window only. Operators should avoid making any changes in the Machine Setup/Driver configuration.



Grid

* Change the Major and Minor Grids when creating contours in Contour Creation, which can be turned off when not in use.

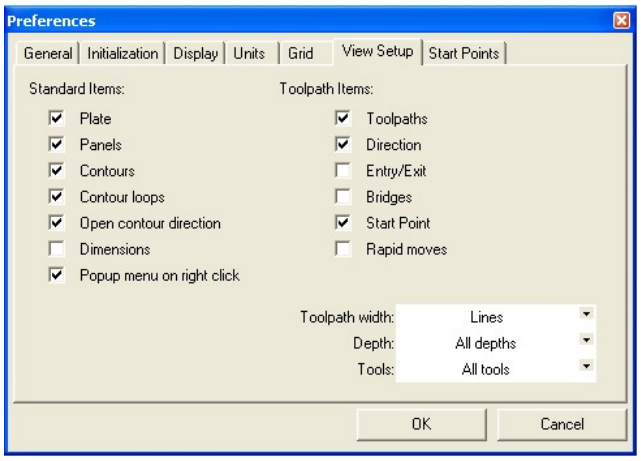
* Review the parameters of the grid. The Interval describes the size of the grid while Size describes the pixel size of the Style.



View Setup

* Adjust View Setup settings to show or hide the listed items when creating contours and tool paths.

* Avoid changing the options in the white window until all paths and options for them have been created. At that point, these settings can be changed to see only certain paths with these 3 options.

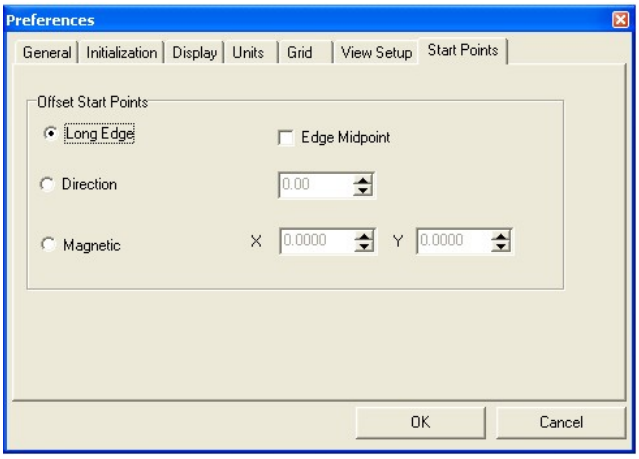


Start Points

* Avoid changing the Start Points settings from the default setting since these offset points can be adjusted manually during the cutting sequence.

* Review the Long Edge configuration as this places the start point on the longest edge of the geometry.

* Select the Edge Midpoint only if the start point should be relocated to the middle of the longest edge.



Software

Innovation. Quality. Performance. Best in Industry. Worldwide.

MultiCam Software

The ***MultiCam*** software installation CD contains the ***MultiCam*** Motion Controller software, which provides the interface between the computer and the ***MultiCam*** Router. The installation CD also contains machine schematics, software drivers, and a routing guide that contains tips on different cutting tools.

Before installing ***MultiCam*** software, customers should make sure there are no other applications open on the computer. Internet Explorer v5.0 or newer is required for this installation.

A. Insert the ***MultiCam*** Installation CD into the CD-ROM drive. This will normally cause the computer to read the CD and then direct the customer through the commands for installation.

B. Follow these steps if the installation does not start automatically:

1. Select the Windows Start button at the bottom left of the screen.
2. Select *Run*.
3. Click *Browse* on the Run Dialog box.
4. Select the Multicam Installation Folder from the directory.
5. Select Setup.Exe.
6. Click *Open*.
7. Click *OK* on the Run dialog box.
8. Follow the setup menus for a complete installation of the ***MultiCam*** software.

Software

Motion Controller Utilities

The ***MultiCam*** software was developed to provide the operator more options and to interface with the newer Windows software. In addition to supporting the DNC function, the ***MultiCam*** software also has options for digitizing, as well as JobPreviewer, JobReporter, bar code scanning, and tool diameter compensation. These tools are all written in 32-bit code and run directly from Windows 2000 or higher. Not all machines have been configured with Suite4, though many of the same features are included.

PSS

- JobServer
- MotionMechanic
- JobPreviewer
- Machine Connections
- JobReporter

Suite4

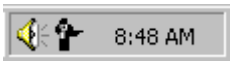
- JobNameServer
- MotionMechanic
- JobEditor
- ConnectionManager
- JobReporter
- JobConsole

Software

JobServer

JobServer is MultiCam's interface between the computer and the ***MultiCam*** Router. JobServer is the PC automation tool for the ***MultiCam*** Motion Controller with specific Control Settings and Project Settings.

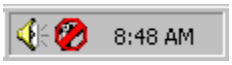
JobServer runs unnoticeably in the Windows system tray on the computer and handles operations between the host computer and the ***MultiCam*** Router.



JobServer was designed to handle DNC and Digitize without the user having to monitor the host computer. While JobServer handles HotWatch and JobQueue operations, HotWatch must be activated by the operator, and JobQueue requires the operator to select job files to queue and then start the created queued files.

JobServer will generally be started by a shortcut in Windows Start/Programs/StartUp folder. This shortcut will be created during installation of the software.

JobServer will automatically release the configured Com Port and enter a sleep mode when another program accesses the port. This allows JobServer to remain running while other programs are used to access the controller. When in sleep mode, the system tray icon will appear like this:



When JobServer is not in sleep mode, DNC and Digitize are always active and waiting for the controller to request them. HotWatch can be configured to always watch the Inbox. The only component that requires user interaction on the host computer is JobQueue.

Advanced Information

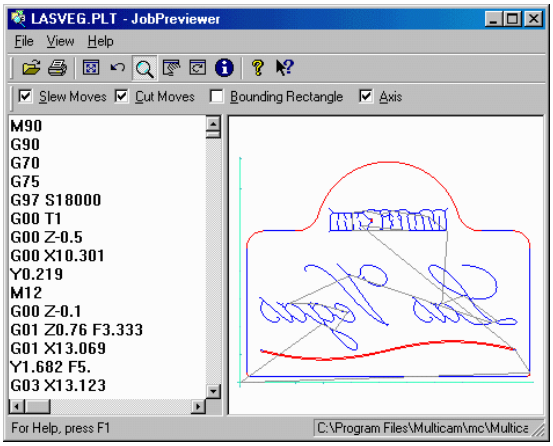
"C:\MultiCam\MC\JobServer.exe" /C="Unknown"

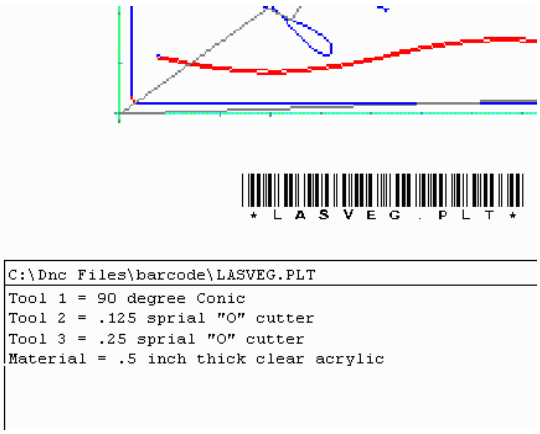
If the JobServer path has spaces in it, the operator must be sure to quote the path as shown above. The /C= switch denotes the Connection Name. If the Connection Name has spaces in it, the operator must be sure to quote the Connection Name also. Software

Innovation. Quality. Performance. Best in Industry. Worldwide.

JobPreviewer

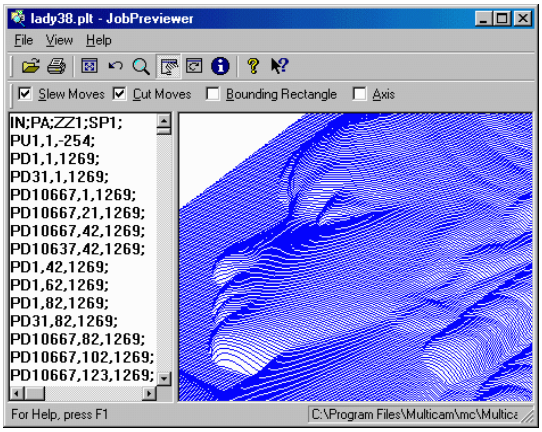
The ***MultiCam*** JobPreviewer allows the operator to preview the job before it is sent to the router table. This is an excellent way to verify the G Code or HPGL Code before the file is cut. The preview will simulate the machine code and display a graphic representation to the operator. To assist with job optimization, the operator can display the machine slew moves to verify the efficiency of the file. The program will also count the number of slew moves, arcs, and lines in a cut file.




* L A S V E G . P L T *
C:\Dnc Files\barcode\LASVEG.PLT
Tool 1 = 90 degree Conic
Tool 2 = .125 sprial "O" cutter
Tool 3 = .25 sprial "O" cutter
Material = .5 inch thick clear acrylic

JobPreviewer can also be used to produce shop floor work orders. Printed work orders can be used to communicate with customers and employees as well as providing special instructions for the work. The work order contains a graphical representation of the cut file, tool specifications, material descriptions, and user notes. This ability of the JobPreviewer can further improve shop floor management if the optional bar code scanning interface is added. The operator will be able to scan the bar code on the bottom of the work order at the machine, and DNC will automatically access the correct job and begin the cutting sequence.

JobPreviewer supports full 3-axis milling and can be used to view the 2D- and 3D-tool path before the job file is processed on the machine.

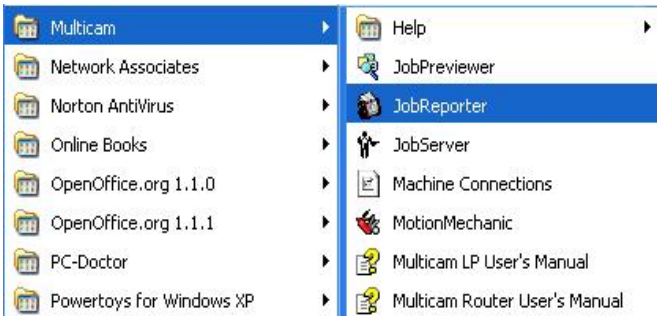


Software

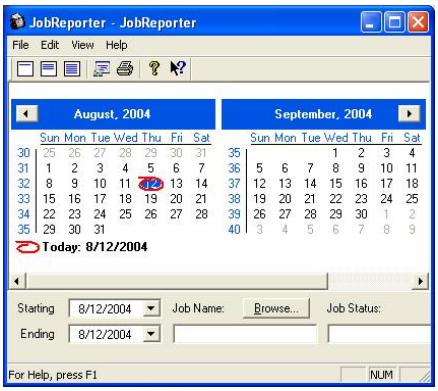
JobReporter

The JobReporter interface analyzes the data collected by DNC, HotWatch, and JobQueue from a simple calendar interface. Daily, weekly, monthly, and yearly reports can be produced. Reports are broken up into multiple work shifts and can even track a specific job. Once the report is created, the data can be exported using many popular spreadsheet or database formats.

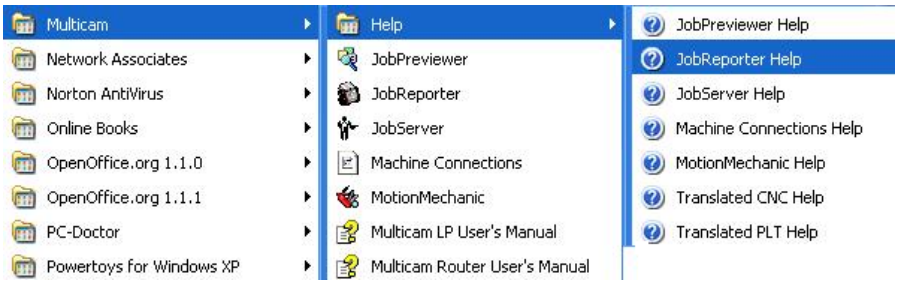
1. Click Start/Programs/MultiCam/Job Reporter to access the Job Reporter function from the main JobReporter screen.



2. Click *Help* on the tool bar or press F1 for detailed instructions once the JobReporter program has been opened.



3. Click Start/Programs/Multicam/Help/JobReporter Help to launch help directly from the ***MultiCam*** folder.



Software

MotionMechanic

The MotionMechanic application gives the operator advanced technical control over the setup of the motion controller from a host PC with several specialized functions. Operators can verify that MotionMechanic is active on the PC by locating the MotionMechanic icon in the Windows system tray at the bottom left.

* **Terminal Emulator** provides direct access to the controller for development, diagnostics, or troubleshooting.

* **Text File Editor** views, edits, or develops files related to the controller on the host PC. Files to be edited include job files, configuration files, and any ASCII text file.

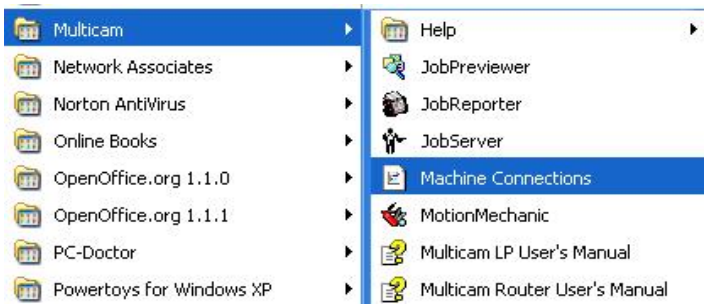
* **Session Log** keeps a record of communication error statistics used for troubleshooting communications problems. Session logs are stored between sessions of MotionMechanic.

Software

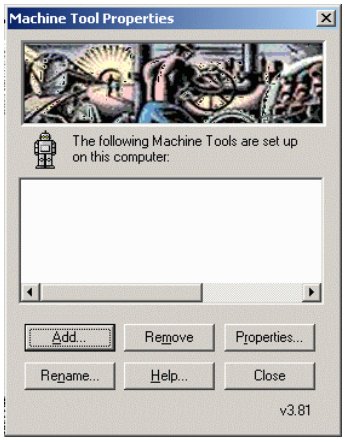
Machine Connections

The Machine Connections applet is located under the ***MultiCam*** folder and in the Windows Control Panel.

1. Click Start/Programs/MultiCam/Machine Connections to access the Machine Connections applet.



2. Select *Help* for a detailed description of each button in the main Machine Tool Properties applet.



3. Click Start/Programs/Multicam/Help/Machine Connections Help to launch the Help function directly from the ***MultiCam*** folder.

Software

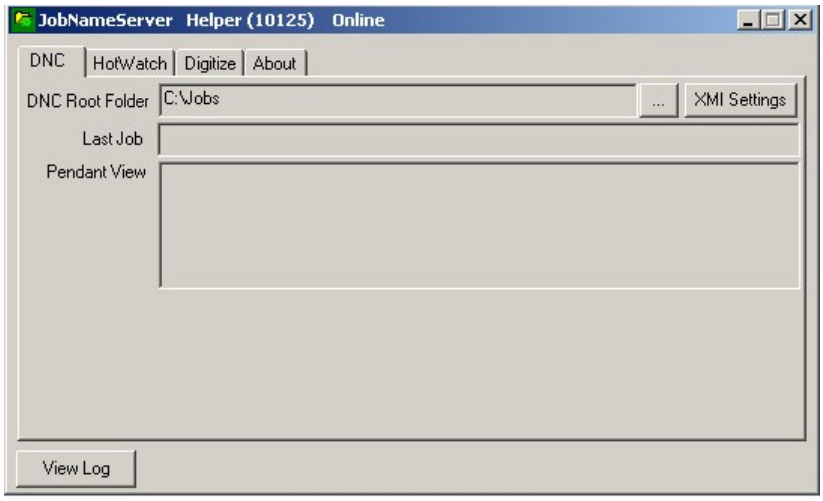
Suite4 Motion Controller Utilities

The Suite4 contains some additional features for standard PSS options and additional options with more advanced features. Operators can access additional information on these features in C:/Program Files/Machine Tools Suite4.

Software

JobNameServer

The JobNameServer program connects the operator with folders on the network or host PC by finding jobs quickly and executing them from the keypad. Other features of JobNameServer can help automate the production process. Operators can click on the tab and start a program or learn about the system.

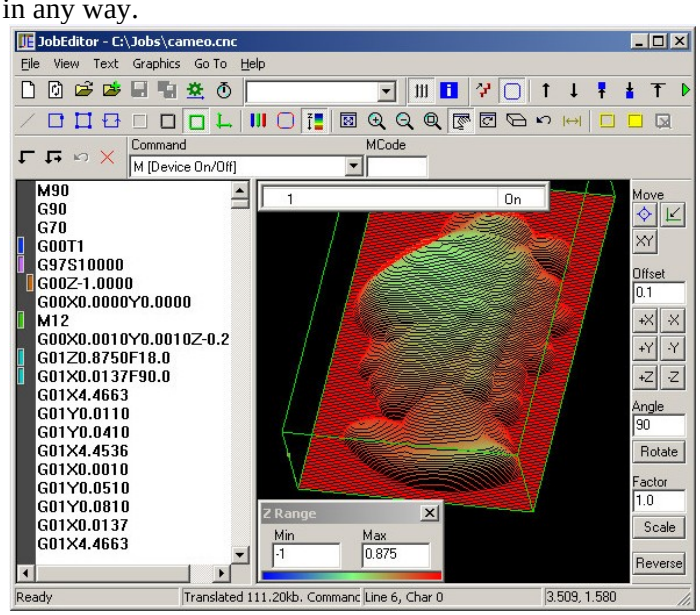


a. **DNC** allows the operator to select and execute jobs from the network or host PC using either a keypad or touch screen interface.
b. **HotWatch** helps automate the production process by monitoring a folder on the network and automatically executing jobs that are placed into that folder.
c. **Digitize** helps operators reproduce 2D and even 3D objects. Not all machines support Digitize, and operators can contact ***MultiCam*** for additional information. A machine supporting Digitize uses special sensors to scan the object and create a job file based on the object. Once the object has been digitized, the job file can be executed as many times as necessary
d. **About** shows the versions of JobNameServer and the associated files used by JobNameServer.

Software

JobEditor

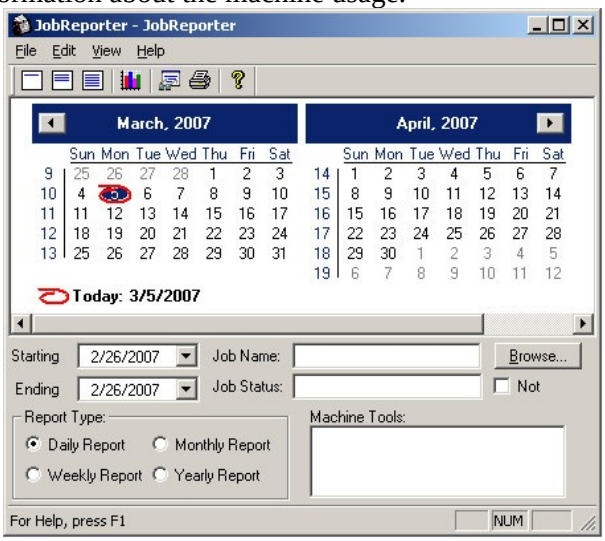
The Job Editor application allows operators to preview and edit jobs through resizing, repositioning, or modifying current job files in any way.



Software

JobReporter

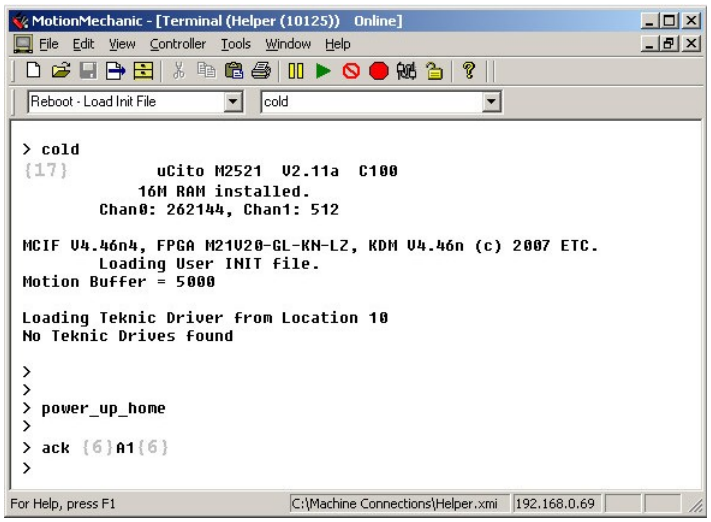
The JobReporter generates reports on machine use. The report results can be filtered and modified to obtain general or specific information about the machine usage.



Software

MotionMechanic

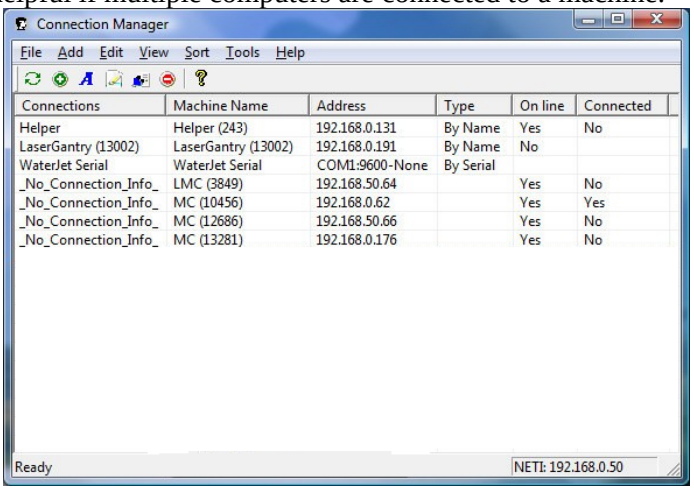
The MotionMechanic application provides the operator with advanced technical control over the use and setup of the motion controller from a host PC.


MotionMechanic - [Terminal (Helper (10125)) Online]
File Edit View Controller Tools Window Help
Reboot - Load Init File
cold
> cold
{17} uCito M2521 V2.11a C100
16M RAM installed.
Chan0: 262144, Chan1: 512
MCIF V4.46n4, FPGA M21V20-GL-KN-LZ, KDM V4.46n (c) 2007 ETC.
Loading User INIT file.
Motion Buffer = 5000
Loading Teknic Driver from Location 10
No Teknic Drives found
> power_up_home
> ack {6}A1{6}
For Help, press F1
C:\Machine Connections\Helper.xmi
192.168.0.69

Software

ConnectionManager

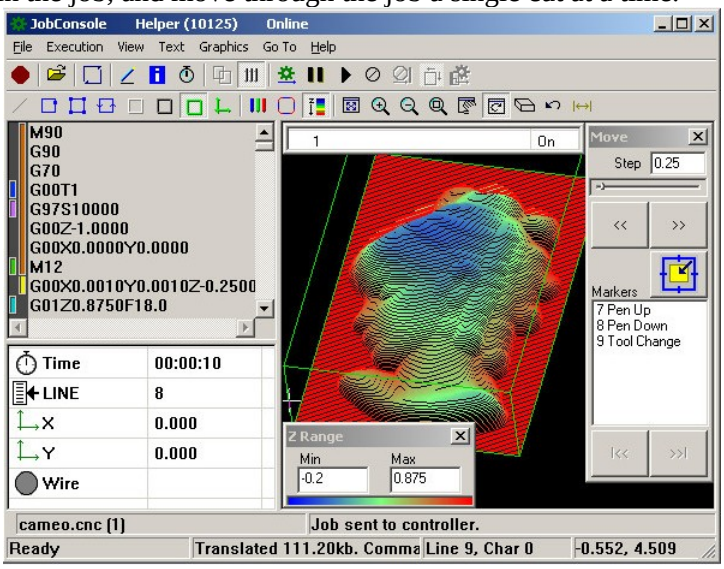
The ConnectionManager helps in creating and configuring the connection between a host PC and the machine tool and has built-in testing features for diagnosing communications issues and sending email correspondence for assistance. Operators can also establish one particular computer to be in charge of all communications, which is helpful if multiple computers are connected to a machine.


Connection Manager
File Add Edit View Sort Tools Help
Connections | Machine Name | Address | Type | On line | Connected
Helper | Helper (243) | 192.168.0.131 | By Name | Yes | No
LaserGantry (13002) | LaserGantry (13002) | 192.168.0.191 | By Name | No
WaterJet Serial | WaterJet Serial | COM1:9600-None | By Serial
_No_Connection_Info_ | LMC (3849) | 192.168.50.64 | Yes | No
_No_Connection_Info_ | MC (10456) | 192.168.0.62 | Yes | Yes
_No_Connection_Info_ | MC (12686) | 192.168.50.66 | Yes | No
_No_Connection_Info_ | MC (13281) | 192.168.0.176 | Yes | No
Ready
NETI: 192.168.0.50

Software

JobConsole

The JobConsole program acts as the main graphical interface between a machine tool and the operator, allowing the operator to preview and execute jobs as well as monitor the job progress, pause and resume the job, move to a specific point in the job, and move through the job a single cut at a time.



Software

JobServer Digitize

The Digitize component of JobServer allows the operator to Jog the machine around and trace a pattern to record movements that will be played back later, thus simplifying new job file creation, with several Digitize Commands.

Using the Digitize functions allows the operator to create a G-Code (CNC) job file by tracing a pattern. Once the pattern is traced, the job file can be saved and reviewed. Once the file is saved, the job is easily executed using DNC. Digitize is only operable with a ***MultiCam*** Router equipped with the Edge Scan initialization file load in the controller. Software

JobServer HotWatch

The HotWatch component of JobServer allows job files to be automatically transferred to the ***MultiCam*** Router by monitoring a specified directory on the computer. Whenever a file with a known extension appears in this Inbox directory, HotWatch will transmit it to the router and then move the file from the Inbox directory to the Outbox directory. These directories do not have to be located on the local hard drive but can exist as shared folders on a local or wide area network. HotWatch can be configured to check for new files at specified intervals to help minimize network traffic.

The full path of the Inbox and Outbox directories is displayed in the HotWatch applet, and these directories can be changed at any time by clicking on the Browse button to select another path. The HotWatch Status informs the operator of all activities that HotWatch is performing.

HotWatch does not automatically monitor the Inbox directory or transmit job files. Operators must manually set up HotWatch and activate this function. Software

JobServer JobQueue

The JobQueue component of JobServer allows jobs for production runs to be created and set up. The job file must be created and started before JobQueue will execute, and the operator can set up JobQueue in the JobQueue window. Once these settings are made and JobQueue is started, the operator will be prompted to start each job after seeing the repetition number of the job being executed. This allows the operator to move to another home position or unload and load material before the cutting sequence begins. Software

JobServer - Control Settings

When ***MultiCam*** software is installed, JobServer's control settings will be configured for each host computer. The Machine Tools Control Panel applet is used to set up and maintain connections between the PC and the controller of the ***MultiCam*** Router. Any current setup connection will be displayed.

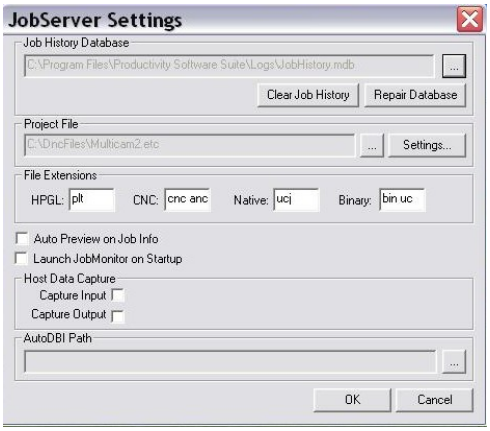
In JobServer, the Machine Tool name is used on the command line to tell JobServer what machine to connect to for the proper settings. This allows JobServer to be opened more than once and communicate to a different machine. To edit a setup connection, the operator must highlight the connection and select *Properties*. The operator may select *Remove* to delete the entry and then select *Add* to add a new connection.

Software

Innovation. Quality. Performance. Best in Industry. Worldwide.

JobServer - Project Settings

The JobServer Project Settings dialog can be found by right-clicking on the JobServer icon in the Windows system tray. After selecting the Program Setting, the operator can adjust the settings to show individual preferences as well as particular settings required by ***MultiCam***. Some of the settings are not used by every ***MultiCam*** operator, and operators can contact ***MultiCam*** Tech Support for answers if they are unsure about their requirements.



The Job History Database is used by all of the components of JobServer to record activity. When files are executed, the file name, date, start time, completion time, total job time, and final outcome of the job are all recorded in this database. Job completion, whether the job was cancelled or an error occurred, is recorded in this file. The database is in the Microsoft Access MDB format. The complete job history can be permanently removed by clicking on the Clear Job History button. When the database becomes very large and starts slowing down, operators can run maintenance on the system by clicking on the Repair Database button.

Certain information, such as M Codes and other CNC- or HPGL-specific information, is stored in Project Files. Operators can select a Project File from Program Settings. Creating a new file or changing values in an existing file can be done by selecting the Settings button next to the project filename. More information on the Project settings is provided in the MotionMechanic documentation.

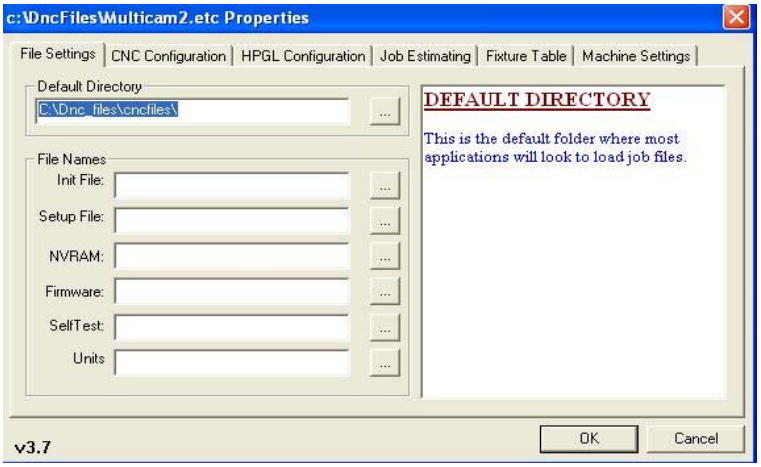
File extensions that JobServer will consider as executable jobs are listed under the File Extension section. File extensions and the corresponding languages are displayed. Using DNC, the operator can view the various job files that correspond to the file extensions on the display. HotWatch will only detect and run files corresponding to the displayed file extensions. Operators must enter the appropriate file extensions in the associated edit box for that language. Separating extensions with a space will specify multiple file extensions for a given language.

One type of language that needs explanation is Binary. When a job is sent to the controller with the extension listed in the Binary box, the job is sent in binary format; therefore, no processing will be done with the job before it is sent. This is useful for Raster jobs, which contain binary information.

For diagnostic reasons, the RS232 Data Capture options are provided. If one of the selection boxes is checked, a filename will be requested, and the system will store data under that filename exclusively until the selection box is unchecked. This procedure should only be used for diagnostic purposes. Because these capture routines are capturing from and to a controller, the port must be available for these options to take effect. If the JobServer icon has the red line through it, then the port is unavailable, which would prevent the operator from setting these specific options. Software

File Settings

The File Settings function of the Project Settings allows the operator to search through files and folders on the system to locate specific job files and extensions after identifying the Default Directory and selecting the file name.



If the Project Settings option is selected without a project open, the Open Project dialog box will open and prompt the operator to select a project before beginning the cutting sequence. At this point, the operator may select the Project Settings option and proceed with the job file.

A. Default Directory displays the path currently specified for the application to follow when searching for files with the .uc extension. The operator can browse the drive(s) of the host PC to locate the appropriate directory with the Browse button to the right of the edit box ... . Once the correct path has been determined and the appropriate directory has been located, the operator may select *OK*.

B. File Names consists of several edit boxes, each specifying the path and file name of the appropriate file. The operator can browse the drive(s) of the host PC to locate the appropriate directory with the Browse button to the right of the edit box ... . Once the correct path has been determined and the correct file is displayed in the dialog box, the operator can either highlight the file of interest and select Open or double-click the file to open it automatically.

1. *Init File* of extension .uc is the initialization file of the application.

2. *Setup File* is of extension .htm.

3. *NVRAM* is of extension .uc.

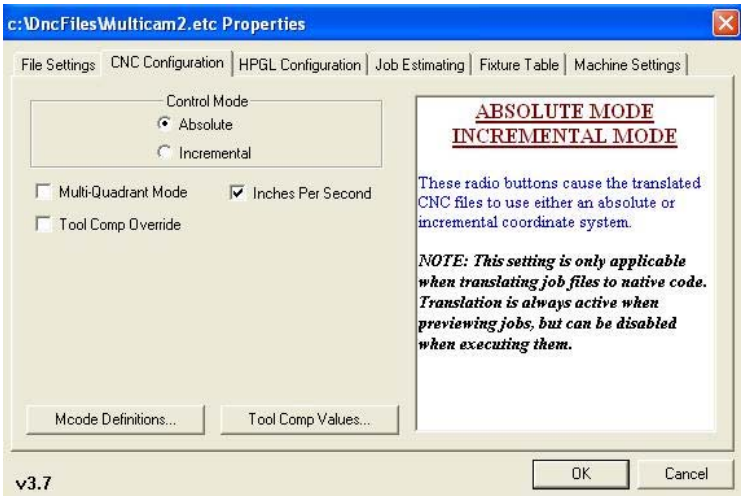
4. *Firmware* of extension .bin is the application interpreter with the controller with files written in assembler language.

5. *SelfTest* is of extension .uc, .ini, .txt.

6. *Units* is of extension .uc, .ini, .txt.

CNC Configuration

Project Settings offer several different modes for working with M and G Codes. If the Project Settings option is selected without a project open, the Open Project dialog box will open and prompt the operator to select a project before beginning the cutting sequence. At this point, the operator may select the Project Settings option and proceed with the job file.



The operator must select absolute or incremental control mode to begin the cutting sequence. Additional classifications and parameters can be set once the control mode is defined.

A. Control Mode

1. Absolute mode (G90) instructs the controller system to determine measurements from the established Soft Home.

2. Incremental or relative mode (G91) instructs the controller system to determine measurements from the last tool position.

B. Multi-Quadrant Mode refers to G74 (multi-quadrant mode off) and G75 (multi-quadrant mode on) codes. When this option is checked, the controller will function in multi-quadrant mode. When this option is unchecked, the controller will function in single-quadrant mode.

C. Mcode Definitions

1. *Add* opens a dialog box with several combination boxes, allowing the operator to establish an M-Code definition with several parts.

2. *Mcode* shows the code designation.

3. *Device* designates a numeric value for each associated tool.

4. *State* indicates whether the system is active (with tool engaged) or inactive (with tool disengaged).

5. *Graphic* indicates either Yes (the sequence typically follows a tool function) or No (the sequence follows non-tool functions).

6. *Edit* allows the operator to edit or modify existing M-Code definitions by selecting the M Code of interest and modifying the information.

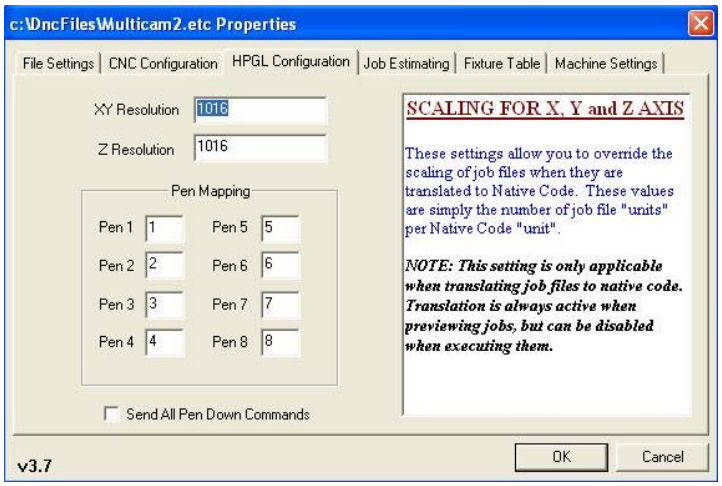
7. *Delete* allows the operator to remove an existing M-Code definition by selecting the M Code of interest and deleting the information.

D. Tool Comp Values allow operators to preview jobs and can be used instead of flash parameters when JobPreviewer is run without a controller or without MotionMechanic being installed. Tool Comp Values match the parameters of the job file. Each tool has its own tool compensation distance, and up to 10 values can be set to compensate for the tool width when cuts are being made. Software

HPGL Configuration

The HPGL Configuration screen is set up for inches with a resolution setting of 1016. This configures the controller for a 1” to 1” scale. Operators using metric units should set the resolution to 40.

The resolution should be checked when the part size is different from the programmed part size. The white box to the right of the settings provides additional information and help for whichever function is selected.



Software

HPGL Command Support

The ***MultiCam*** control system internally translates HPGL. Feedrate, Depth, and Tool Lift can be set by the file or by the machine's keypad interface. If the Feedrate, Depth, and Tool Lift are set at the machine prior to the file being sent, the file will override all those values set at the machine. If no value is sent in the file, then the machine will use the value programmed at the keypad. The control system ignores the HPGL VS command.

At the beginning of the file, the controller **must** be sent the IN command to begin processing plot commands. If the letter I is **not** the first character sent, then a linefeed or semi-colon must be sent to re-initiate the sequence. A semi-colon must follow the IN command. If any other character is sent during this sequence, then the complete sequence must be restarted. This command sequence must be sent again if an SP or SP0 command is sent. The recommended initialization string is ";IN;" and the recommended termination string is "SP0;". All files should be written in absolute mode. Some of the early routing systems do not support relative arc commands.

HPGL Resolution: 1016 inch =40 Metric

The following pages contain all of the supported commands. Bracketed information is optional, while braced information is exclusive. A semi-colon or a linefeed **must** terminate all commands, and spaces **cannot** be substituted for required commas in the syntax of a command. All Z-axis arguments are only valid if the 3D mode is set. All commands are integers unless otherwise specified.

Help documents on the HPGL and CNC systems are located in the ***MultiCam*** folder, and operators can access this folder through either Start/Programs/Multicam/Help/Translated PLT Help or Start/Programs/ Multicam/Help/Translated CNC Help.

Example Files

Multi Tool File

```
;IN;PU;ZZ0;
PA0,0;
TCTool 1;
ZD127;ZU400;
PA1016,1016;PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
PU;TCTool 2;
ZD127;ZU400;
PA2032,2032;PD2032,2032;PD10160,2032;
PD10160,10160;PD2032,10160;PD2032,2032;
PU;TCTool 3;
ZD127;ZU400;
PA3048,3048;PD3048,3048;PD9144,3048;
PD9144,9144;PD3048,9144;PD3048,3048;
PU;PA0,0;
SP0;
```

Automatic Tool Changer File (or Multi Head)

```
;IN;PU;ZZ0;
PA0,0;
SP3;
ZD127;ZU400;
PA1016,1016;PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
PU;SP1;
ZD127;ZU400;
PA2032,2032;PD2032,2032;PD10160,2032;
PD10160,10160;PD2032,10160;PD2032,2032;
PU;SP2;
ZD127;ZU400;
PA3048,3048;PD3048,3048;PD9144,3048;
PD9144,9144;PD3048,9144;PD3048,3048;
PU;PA0,0;
SP0;
```

Arc File

```
;IN;ZZ0;SP1;
PA1360,2450;ZD0.500000;PD;
AA1270,2540,360.000000;PU;
PA2010,3048;PD;PA1292,3766;PU;
PA2630,3720;PD;AA2540,3810,360.000000;
PU;PA3290,4263;PD;PU;PA4154,3720;
PD;AA4064,3810,360.000000;
PU;PA4902,4263;PD;PU;PA5678,3720;
PD;AA5588,3810,360.000000;PU;
PA4064,2921;PD;PA2540,2921;
AA2540,2540,180.000000;PA5588,2159;
AA5588,2540,180.000000;PA4064,2921;
PU;PA4154,1180;PD;AA4064,1270,360.000000;
PU;PA5678,1180;PD;AA5588,1270,360.000000;
PU;PA6948,2450;PD;AA6858,2540,360.000000;
PU;PA2630,1180;PD;AA2540,1270,360.000000;
PU;PA7747,2540;PD;AA5588,2540,-90.000000;
PA2540,381;AA2540,2540,-
180.000000;PA5588,4699;
AA5588,2540,-90.000000;PU;PA8128,0;SP0;
```

Sample 3D file

```
IN;ZZ1;PU;SP1;
PA5000,5000,-200;
PD5000,5000,500;
PA5000,15000,1000;
AA10000,15000,-180;
PA15000,5000,750;
AA10000,5000,-180;
PA5000,5000,-200;
PU;SP0;
```

Multi Pass File without Lifting the Z

```
;IN;PU;
PA0,0;
SP1;
ZD100;
PA1016,1016;
PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
ZD200;
PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
ZD300;
PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
ZD400;
PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
PU;SP0;
```

HPGL Commands

Initilization

IN instructs the controller to begin processing the HPGL plot file. Without this, the commands in the file are received but never executed. If multiple IN commands are found during the execution of the file, the controller performs a Pause/Cancel operation. All motion from the previous job that has yet to be executed is lost, and the new information is executed. When the IN command is executed, the HPGL 3D mode is cleared. The 3D mode can be reset using the ZZ command. A PU command will be executed before the system executes the first HPGL move command.

ZZ (0,1) establishes that a value of 0 sets 2D mode and a value of 1 sets 3D mode. In 2D mode, all commands must have 2 axes, while all commands must have 3 axes in 3D mode.

Motion

PA[X,Y][,Z] moves to an absolute HPGL position and sets absolute mode for future PU and PD commands. If no arguments follow the command, only absolute mode is set.

PD[X,Y][,Z] executes a Pen Down then moves to the requested position, if one is specified. This position is dependent on whether absolute or relative mode is set. The Feedrate Override knob is turned ON.

PU[X,Y][,Z] executes a Pen Up then moves to the requested position, if one is specified. This position is dependent on whether absolute or relative mode is set. The Feedrate Override knob is turned OFF.

AA X Ctr,Y Ctr,Angle is a floating point requiring a non-integer value that draws an arc with its center at X,Y. A positive angle creates a counter-clockwise arc.

ZA Z moves the Z-axis to the absolute position specified. This feature is for both 2D and 3D modes.

Speed

SF[XY-rate],[Z-rate], floating points, sets the feedrate for the machine tool during Pen Down. All rates are in mm/sec. The Z rate applies to the plunging feed rate of the Z-axis. In 3D mode, the Z-axis rate is not used.

Other

SP[0, 1, 2, ..., 8...] selects a new tool for use. If zero or no pen number is given, the controller performs an end of file command (EOF). Once an EOF is performed, no motion is executed until a new IN command is received. Pens 1—8 are spindles, and pens 11—12 are drills.

ZD Z, floating point, sets the distance from the surface for the Z-axis to move down on a PD command (Depth). Changing the ZD value will change the depth on all following PD commands. The value set with this command is **not** used when running in 3D mode.

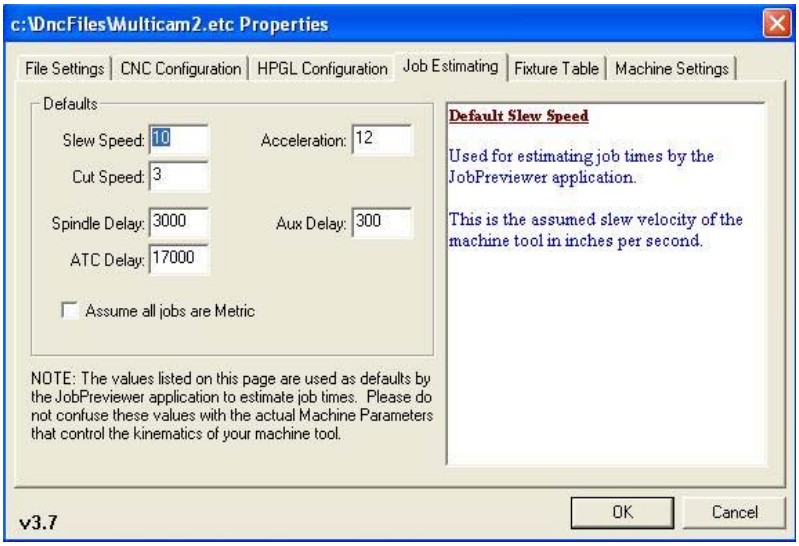
ZU Z, floating point, sets the distance from the surface for the Z-axis to move up on a PU command (Tool Lift). Changing the ZU value will change the lift distance on all following PU commands. The value set with this command is **not** used when running in 3D mode.

TC [text] issues a tool change command. The text following the TC command will appear on the keypad display at the appropriate time in the tool change sequence. The operator will have the ability to surface the new tool, change speeds, depths, and the lift. If this information is going to be set by the file, the appropriate commands must be sent after the tool change command. Software

Job Estimating

The Job Estimating screen provides operators with a place to enter values for the JobPreviewer function to use when estimating Job times. **These settings have no effect on the actual machine operation.**

The White box to the right of the settings provides help for whichever function is selected.


c:\DncFiles\Multicam2.etc Properties
File Settings
CNC Configuration
HPGL Configuration
Job Estimating
Fixture Table
Machine Settings
Defaults
Slew Speed: 10
Acceleration: 12
Cut Speed: 3
Spindle Delay: 3000
Aux Delay: 300
ATC Delay: 17000
Assume all jobs are Metric
NOTE: The values listed on this page are used as defaults by the JobPreviewer application to estimate job times. Please do not confuse these values with the actual Machine Parameters that control the kinematics of your machine tool.
Default Slew Speed
Used for estimating job times by the JobPreviewer application.
This is the assumed slew velocity of the machine tool in inches per second.
v3.7
OK
Cancel

Software

MultiCam M- and G-Code Support

The following table lists the supported G and M Codes for the *MultiCam* Controller. Parameters within brackets are optional. The fields represented by "d.d" may be any decimal number, and fields represented by "d" may be any positive integer. All lines of machine code must start with either an M Code or a G Code.

G00 [Xd.d] [Yd.d] [Zd.d] [Fd.d] [Td] [Ctext string] - High speed move (slew)
G01 [Xd.d] [Yd.d] [Zd.d] [Fd.d] - Linear move (machine)
G02 [Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d] - CW 2D circular move
G03 [Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d] - CCW 2D circular move
G04 Fd.d - Dwell (seconds)
G17 - Specify XY plane for helical
G18 - Specify ZX plane for helical
G19 - Specify YZ plane for helical
G37 - Find home
G62 - Clear soft home
G70 - English programming (inches)
G71 - Metric programming (mm)
G72 [Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d] - CW 3D circular move
G73 [Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d] - CCW 3D circular move
G74 - Incremental mode for G02/03 arcs
G75 - (G90/G91) mode for G02/03 arcs
G83 Rd.d Zd.d Dd.d [Fd.d] - Peck drill
G90 - Absolute coordinate mode
G91 - Incremental coordinate mode
G92 [Xd.d] [Yd.d] [Zd.d] - Set soft home
G97 Sd - Set spindle speed (rpm)
M00 - Program pause
M01 - Optional pause
M02 - Program end
M11 - 2D device on (like HPGL PD)
M21 - 2D device off (like HPGL PU)
M12 - 3D device on
M22 - 3D device off
M30 - AUX 4 on not affected by the Pause button
M31 - AUX 4off not affected by the Pause button
M90 - Program start
M99 - Exit CNC interpreter

The following table lists the letters used to denote various arguments in ETC CNC version 1.0.

C - Tool change operator message (used in G00)
D - Peck drill delta (used in G83)
F - Feed rate (used in G00, G01, G02, G03, G72, G73, G83)
F - Dwell (used in G04)
G - Preparatory function
I - Circular interpolation value in X dimension (used in G02, G03, G72, G73)
J - Circular interpolation value in Y dimension (used in G02, G03, G72, G73)
K - Circular interpolation value in Z dimension (used in G02, G03, G72, G73)

M - Miscellaneous function (control function)
N - Sequence number
R - Beginning Z motion dimension (used in G83)
S - Spindle rpm (used in G97)
T - Tool change (used in G00)
X - X motion dimension
Y - Y motion dimension
Z - Z motion dimension

The following is a 5” square with a rapid level of 0.5” above the material, feed down at 100 ipm, cut feed at 200 ipm, rapid down to 0.1” above the material, and a cut depth of 0.25”.

```
M90
G90
G70
G75
G00 T1
G00 Z-0.5
G00 X0. Y0.
M12
G00 Z-0.1
G01 Z0.25 F1.667
G01 X5. F3.333
G01 Y5.
G01 X0.
G01 Y0.
G00 Z-0.5
M22
G00 X0. Y0.
M02
```

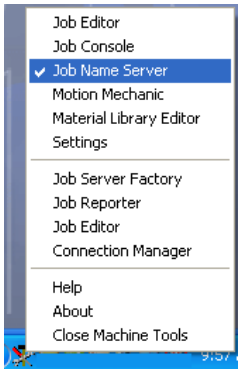
The following is a 5” circle clockwise, with a center at 2.5,2.5, rapid level of 0.5”, feed down at 60 ipm, cut feed at 120 ipm, rapid down to 0.1”, and a cut depth of 0.4”.

```
M90
G90
G70
G75
G00 C1/2”bit
G97 S18000
G00 Z-0.5
G00 X2.5 Y2.5
M12
G00 Z-0.1
G01 Z0.4 F1.
G02 I2.5 J2.5 F2.
G00 Z-0.5
M22
G00 X0. Y0.
M02
```

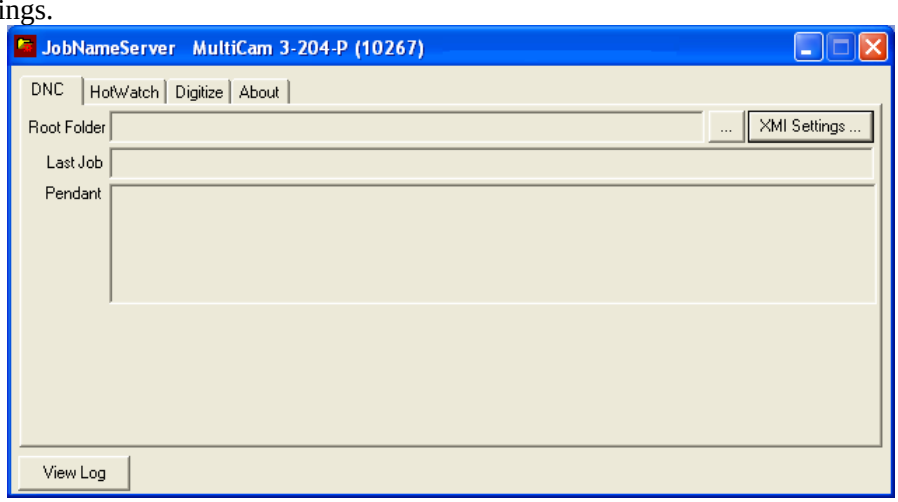
Mapping M Code to Output

Operators can add M Codes to an output on their existing machines, assign the code to a particular device, and set the active status by completing the following steps:

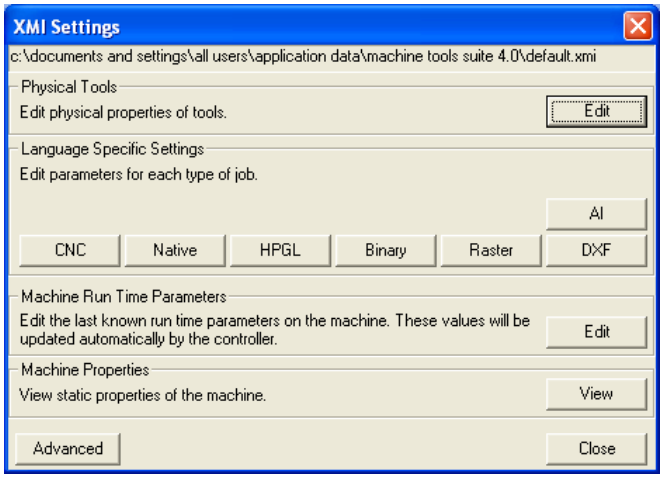
1. Open MachineTools and select JobNameServer.



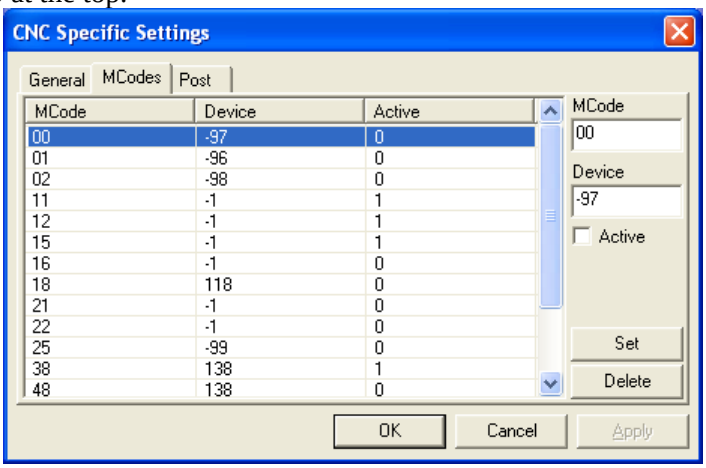
2. Select XMI Settings.



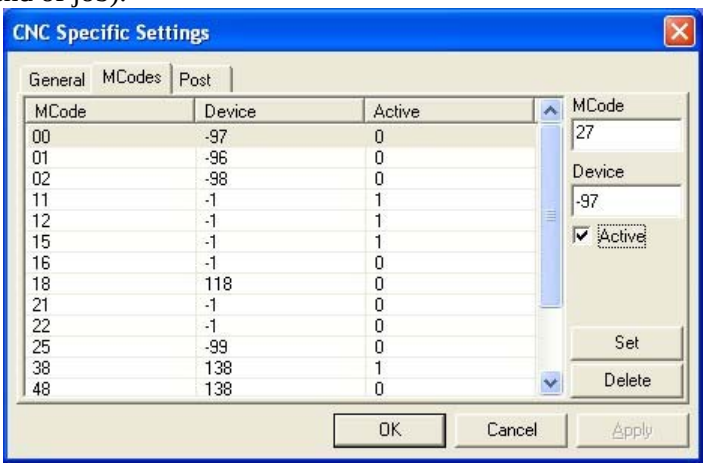
3. Select CNC.



4. Click on the MCodes tab at the top.



5. Enter the M Code (e.g., 27), correct the device as needed, set the status to Active by clicking in the check box, and select *Set*. **Active** indicates a motion command (e.g., start spindle) while **Inactive** indicates a non-motion command (e.g., end of job).



Any M Code, including M12, can be mapped per the operator's preference; however, JobConsole treats the M12/M22 coding specifically for the start/end of a contour.

6. Download and store an MCode_Device_Map.uc file to location 3. For assistance on any of these steps, please contact ***MultiCam*** Tech Support.

Operators may also modify or delete M Codes from this same dialog.

Modify - Highlight an existing M Code, make changes (e.g., Device -1 instead of -97), and select *Set*.

Delete - Highlight an existing M Code and select *Delete*.

Glossary

The following words are most commonly used in reference to the router and associated activities.

Acrylic - A type of plastic, or a common material used in sign making.

Aggregate - A type of tool that is not vertical and does not spin in line with the spindle axis. An aggregate tool may also rely on the C-axis for operation (e.g., saw).

Axis - A direction in a coordinate system. ***MultiCam*** Routers have 3 axes: X, Y, and Z.

Axis Motor - A motor that causes motion in a particular axis.

Bar Code Scanner - A device that allows operators to retrieve information encoded in a bar code label so that specific notes and directions are kept in line with the correct job file.

Bearing Car - The bearing on which an axis moves along a rail, also known as a bearing truck. There are 2 bearing cars for each rail.

Bearing Rail - The rail on which the bearing cars ride. There are 2 rails for each axis.

Bite Size - The depth of each pass, used in multipass.

Chuck - A clamping device for holding a drill bit.

Climb Cut - A direction the router moves along a cut to produce a climbing motion. For a closed contour, the climb cut is clockwise on the outside of the cut and counter-clockwise on the inside of the cut.

Collet Wrench - A wrench used to loosen or tighten the cover nut on a spindle or tool holder.

Compression Cutter - A spiral up/down cutter. This cutter is used for high feeds in wood and laminates and is configured with a spiral up at the bottom and spiral down at the top.

Conical Cutter - A V-shaped cutter used in relief carving, engraving, and chamfering.

Conventional Cut - The opposite cut of a climb cut, or closed counter-clockwise on the outside of the cut and clockwise on the inside of the cut.

Cross-over Cable - The type of Ethernet connection that is directly connected to the computer.

Cut Conditions - The most efficient condition for standard cutting or engraving. Cut conditions change the motion parameters of the routing system.

Cut Depth - A Hot Key function that sets the depth of cut. Cut Depth refers to the Z-axis distance the end of the cutter will go below the surface of the material during a cut.

Cut Speed - A Hot Key function that sets the speed of the cut.

Dieline - The path taken by the cutting head to complete the cutting sequence. Dielines are most often used in software programs like Adobe Illustrator.

DNC - The system that allows the controller to access sub-directories on the hard drive of a host PC and is used to transfer files from the PC to the controller.

DNC Log - A listing of all activity completed by the DNC and kept by the DNC.

Drill Attachment - An attachment that allows drilling in addition to the routing spindle.

Dry Run - A controller function that will execute a file with no Z-axis movement. Dry Run is used to show the operator where the tool will go during the cutting sequence.

Emergency Stop - The red mushroom button located on the keypad, as well as on the side of the router, used to remove power from the machine excluding the controller board and limit switches.

External Halt - The keypad display that appears when power is applied to the system with the Emergency Stop button pushed in on the keypad pendant.

Feedrate Override - A Hot Key used to adjust the cutting feedrate of the system while a file is being executed.

Fiducial - A reference mark, or dot, used in MultiVision that allows the CNC machine to recognize the orientation of the material and adjust the cut file accordingly.

Firmware - The operating system of the controller that should only be loaded when instructed by ***MultiCam***. Firmware contains all of the low-level commands and is the first level of controller software.

Flash RAM File - A file containing all of the machine parameters of a particular router. Settings such as table size and resolution are located in the Flash RAM File, which is the third level of controller software.

Gantry - The mechanical part of the router, also known as the bridge, which moves the carriage assembly from the front to the rear of the table.

G Code - The machine code language used by the controller to execute motion commands.

Gearbox- The drive system on Pro series that uses planetary gear reduction to produce rotation of the pinion.

Glide Pad - The attachment for the pressure foot that glides along the material to keep the material in place.

Hard Home Position - The home position determined by the location of the targets and limit switches, typically the front right side of the table.

Head - A reference to a particular cutting tool (i.e., spindle).

Head Mode - A designation for multiple head machines, either All or Auto.

Hot Keys - The one-touch keys on the keypad that perform controller functions.

HPGL - The machine code language that the system executes as a file.

Init File - The second level controller software that contains high-level commands.

JobServer - The motion controller utility program used to communicate between the controller and the host PC.

JobNameServer - The motion controller utility program used to communicate between the controller and the host PC in Suite4.

Kerf - The spacing between parts. The kerf must be larger than the tool diameter.

Keypad - The part of the pendant with the grid of Hot Keys.

Light Curtain - A boundary around the machine created by parallel, infrared light beams that stops the cutting sequence once on object or person crosses the beam.

Linear Tool Changer - The tool changing system with the tool holders lined up along an axis. This is the most common tool changer for the SF series.

Machine Parameters Menu - A tool of the ***MultiCam*** Productivity Software Suite that allows the flash RAM file to be modified.

Menu System - The commands that are not assigned to Hot Keys.

Misting Nozzle - The tip of the flex hose on the misting system.

Misting Unit - The unit that rides on the back of the carriage assembly and provides misting lubrication for cutting metal, available in automatic and manual versions.

Multiple Passes - A cutting movement through material that repeats the contour at different depths with deeper cuts each time based on the bite size.

Multiple Spindles - A router configuration with more than 1 spindle or routing head.

Node - A computer or device that is connected to a Class C or Private network, which is common in most businesses.

Nose Cone Brush - The brush attachment for the pressure foot. The nose cone brush is used when the material is easily scratched by the glide pad.

O Flute - A type of cutter that has a half-moon geometry from the end view.

Operator Station - The stand-alone board separate from the machine and keypad that allows the operator to turn on auxiliary systems or access dual systems if applicable.

Park - The function used in all 3 axes to place the carriage assembly at an established or farthest point away from the material for loading and unloading.

Patch Cable - The type of Ethernet connection that is connected to the computer through a networking component such as a hub, switch, or router.

Pause - The yellow button on the keypad that puts the controller into Pause mode where action is momentarily stopped and can be restarted at the keypad.

Pendant - The control module consisting of the keypad and mechanical inputs.

Pressure Foot - An attachment to the spindle that keeps the material in place and removes chips and other types of debris from the cut material.

Programmable Soft Home - The 9 possible home positions that the operator can store to be recalled at a later time.

Proximity Restart - The controller function that allows the operator to restart a file along any cut move after an interruption such as power outage or tool breakage.

PSS - The ***MultiCam*** Productivity Software Suite (formerly the Router Productivity Suite, {RPS}). This software provides the interface between the computer and the router.

Rack and Pinion - A system of transferring movement from the rotation of the motors to the linear movement of the system by way of gears.

Rotary Tool Changer - The tool changing system with tool holders held on a rotating turret. The rotary tool changer is used on the PRO series.

RPS - The ***MultiCam*** Router Productivity Suite, now referred to as the Productivity Software Suite (PSS).

Sacrifice Material - The material that is placed between the tabletop and the material being cut, also known as spoil board. The sacrifice material is used to protect the table surface and is considered a consumable item.

Self Test - A controller function that allows the system to test itself. The self test is accessed from the menu items and installing the self-test plug on the controller.

Serial Cable - The RS232 cable used to transfer data from the host PC to the controller.

Servo Drive - The servo amplifier used to move the servo motors.

Servo Motor - The axis motor on a servo system.

Side Cover - The covers on each end of the gantry protecting the X-axis motor and limit switches.

Single Flute - A type of cutter that only has 1 cutting edge.

Slew - A non-cutting movement of the system above the material at maximum speed.

Soft Home Position - Any set home position used as the origin for executing programs.

Spindle - The rotating motor that holds the cutting tool.

Spindle Plate - The plate that mounts the spindle to the Z-carriage plate.

Spindle Speed - The speed the spindle rotates, shown in RPMs.

Spiral Downcut - A cutter with flutes that spiral around the shaft. A spiral downcut pushes down the chips that are cut.

Spiral Upcut - A cutter with flutes that spiral around the shaft. A spiral upcut pushes up the chips that are cut.

Spoil Board - The material that is placed between the tabletop and the material being cut, also known as sacrifice material. The spoil board is used to protect the table surface and is considered a consumable item.

Stepper Drive - An amplifier that controls the action of the stepper motors.

Stepper Motor - An axis motor that uses stepper technology.

Straight Flute - A type of cutter with flutes arranged parallel to the center of the cutter shaft.

Surface - The topside of the material used as the Z-axis reference point.

Tool Lift - The Z-axis distance between the bottom of the router and the material. The Tool Lift is also the location of the router during a slew move.

Tool Offset - The distance between the center of the router and the cutting edge. The Tool Offset is also the distance between the desired geometry and the center of the tool path, usually the radius of the router.

Tool Path - The path that the center of the router takes during a cut.

Transmission - The drive system on SF series using a 4:1 belt drive pulley system to produce rotation of the pinion.

Two Flute - A type of cutter that has 2 cutting edges.

uCito - The native language of the controller (pronounced micro-see-tow).

Vector - A line segment showing a particular direction or path.

Vinyl Knife - An attachment that is used to cut vinyl.

Warmup - The process a system goes through to prepare for regular operations (e.g., spindle warmup before any routing begins).

Wood - A common routing material.

X-axis - The axis that refers to the length of the table, front to back.

X Motor Plate - The plate attached to the side of the gantry in which the transmission or gearbox assemblies are bolted.

Y-axis - The axis that refers to the width of the table, left to right.

Y Carriage Plate - The vertical plate behind the Z-axis carriage plate where Z-axis bearing rails are mounted.

Y Motor Plate - The horizontal plate attached to the Y-carriage plate where the Y-axis transmission or gearbox assemblies are bolted.

Z-axis - The axis that refers to the accessible range of the area above the table.

Z Carriage Plate - The vertical plate on the vertical moving part of the carriage assembly where the cutting tool plate is mounted.

Z Motor Plate - The horizontal plate on top of the Y-carriage plate where the Z-axis motor is mounted.

Z Speed - The vertical speed of the spindle assembly or the plunging speed for spindle machines.

Software